



DynaCADD®

Reference Manual

Ditek International

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DynaCADD 2-D Tutorial

This tutorial takes you through the construction of a simple two dimensional part. It is meant to show you how to enter commands in DynaCADD, not to teach professional drawing and drafting practices. In many cases, extra steps were used in order to demonstrate the operation of a greater number of DynaCADD commands. The finished part is provided on disk, in file TUTOR2D.PRT, so that you can compare your results with the actual drawing. The part itself represents a special adjusting screw, with both the front and right orthographic view.

The adjusting screw consists of a shank and a head. Inside the hexagonal head is a cylindrical well that narrows to a smaller diameter. The shank consists of three parts: an upper, middle and lower shank section. To indicate this, without drawing the entire screw, a cutaway area is drawn.

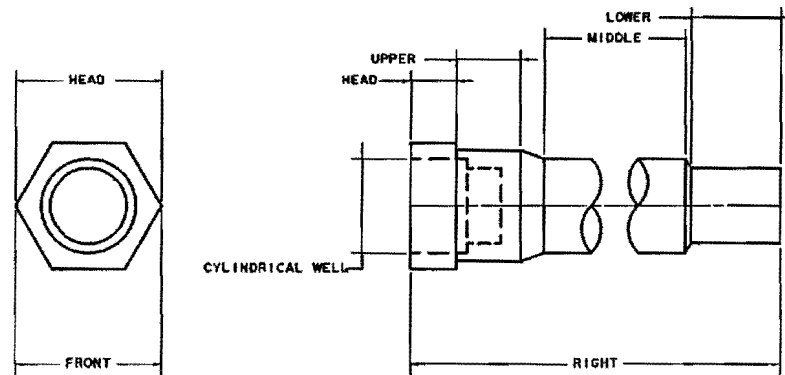


fig. (1) Fully Labelled Adjusting Screw Drawing

Each step along the way is accompanied by a drawing. Try to replicate the drawing as accurately as possible when following this tutorial.

As you progress, some of the explanations and instructions become more concise where the material was explained in a previous step.

As you will soon discover, because the screw is a symmetrical item, you do not need to create the entire right view, only the top half. This is then mirrored to create the bottom.

DynaCADD® 2-D Tutorial

Experiment with the window resizing and zoom commands before starting, because these are useful for examining your drawing in detail as you work. These commands are described in detail in Chapter 4.

In this tutorial, you must select (click on) several icons in the proper order of their command sequence. The names of the commands appear at the upper left of the drawing area window when you move the pointer over the icon. When the proper command appears, click the left mouse button.

In this tutorial we use the term “select” meaning “to activate an option or to choose an entity.” To activate an option, either move the mouse pointer to a button in a dialog clicking the left mouse button or pull down a menu and click on an option. When a dialog button is selected (activated), it appears in reverse type (highlighted). When a menu option such as Inheritance is selected, a check mark appears beside its name in the Tools pull down menu.

Since most commands are formed of several parts, when you activate an icon, additional icon pads reveal the subcommands. You must then find the next icon in the sequence and click on it. Do this until all commands (shown in uppercase) are entered.

When you see the semicolon (“;”), you can either click the right mouse button, or press [;] on the keyboard.

In the command descriptions and screen shots, we use abbreviations to refer to entities and selections:

loc #1, loc #2, etc., refer to locations

ent #1, ent #2, etc., refer to entities.

When you are told to select a location (i.e., loc #1, indicated by a small “+”), look at the illustration and find each location. Move the cross hair to the same location on your screen and click the left mouse button. If you inadvertently select the wrong location, press the delete key to go back a single step and try again. Some locations are actually situated on an entity, and it is important to select a location as close to the indicated end of the entity as possible.

An entity is the basic element within DynaCADD. Entities include such elements as points, lines, circles and arcs. When you are told to select an

entity (i.e., ent #1, indicated by a small "+"), the DynaCADD screen cross hair shows two small boxes (called "traps"). Move the boxes over the desired entity and click the left mouse button. If you inadvertently select the wrong entity, press the DELETE key to go back a single step and try again. Note that, in some commands, you are told to select the same entity more than once.

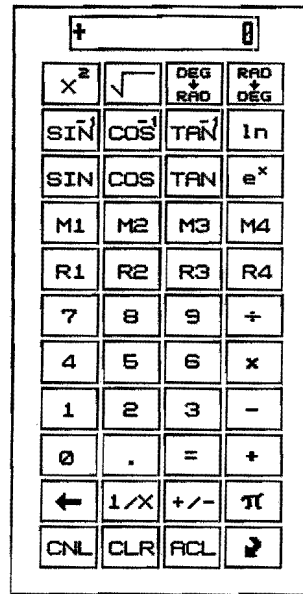


fig. (2) Value Calculator

Some commands produce a pop up scientific calculator where you enter values for specific parameters such as radius or distance. Type in the value or click the pointer over the number icons. Press [Return] or click the Return icon, located in the bottom right hand corner of the value calculator, to enter the value in the calculator window. In the instructions below, the exact values to enter are listed immediately after the icon (command) name.

After you successfully perform each step in this tutorial, you should pull down the File menu and select SAVE PART. This way you can experiment with each finished step, then use the Restore option in the File menu to return to the last saved situation.

For more information on the commands, refer to the main DynaCADD manual. It is particularly useful to read chapters 1 to 4 before starting this tutorial.

Installing DynaCADD - DOS Version

DynaCADD requires MS-DOS version 2.0 or later, 80286/80386/80486, PS/2 or 100% compatible computers, 640K RAM and 1 Mb of EMS memory (EMS 3.2 or 4.0), one 5 1/4" or 3.5" floppy disk drive, a graphics card operating in minimum of 640 X 350 and a mouse. However, we recommend a hard disk drive, math co-processor, VGA high resolution graphics card and at least 2 Mb of EMS memory. On 80386 and 80486 based systems, expanded memory managers such as QEMM from Quarterdeck Office Systems work well with DynaCADD.

The following are the steps taken to install the DOS version of DynaCADD into your system. In the installation program use the up and down arrow keys to move the selection bar. The [Esc] key can be used to back up to the previous dialog.

1. Insert Disk 1 into drive A.
2. Type A: and press [Enter].
3. Type INSTALL and press [Enter].
4. Press [Enter] with the selection bar over the "Install GEM/3 into System" option.
5. Highlight the source drive letter and press [Enter].
6. Highlight the drive on which you wish GEM/3 to be installed and press [Enter].
7. Highlight the appropriate graphics card with the selection bar and press [Enter].
8. Highlight the mouse or tablet type with the selection bar and press [Enter].
9. A menu may pop up asking which serial port the mouse or tablet is hooked up to. Highlight the appropriate port and press [Enter]. The installation program will now proceed to copy the GEM/3 run-time and the suitable screen driver to your hard disk. When it is completed a message will appear.
10. Move the selection bar to "Install DynaCADD into System" and press [Enter].

11. Type in your name and address pressing [Enter] at the end of every line. Press [Enter] through all five lines.
12. Highlight the source drive letter and press [Enter].
13. Highlight the drive on which you wish DynaCADD to be installed and press [Enter].
14. INSTALL will now copy the contents of Disk 1 to your hard disk.
15. When INSTALL is finished with Disk 1, it will ask for Disk 2. Insert Disk 2 into drive A and press [Enter]. INSTALL will now copy the contents of Disk 2 to your hard disk.

DynaCADD is now installed on your system.

Loading DynaCADD - DOS Version

To load DynaCADD:

At the DOS prompt enter the following commands:

1. C: and press [Enter].
2. CD \ and press [Enter].
3. DYNACADD and press [Enter].

Note: During installation, the INSTALL program created three files called DYNACADD.BAT, FONTEDIT.BAT and MAKEPLOT.BAT in the root directory of drive C. These batch files can be moved into any directory that is contained in the system PATH.

Installing DynaCADD - AMIGA Version

DynaCADD requires at minimum an Amiga with 2 Mb of RAM and a hard disk to operate properly. If your system contains a 68020/68030 microprocessor and either a 68881 or 68882 math co-processor, the InstallHD program will copy a version of DynaCADD to your system that takes full advantage of these processors.

DynaCADD® 2-D Tutorial

Since a screen resolution of at least 640 by 400 is required, we strongly recommend the Commodore 2320 de-interlacer or the MicroWay Flicker Fixer for use with DynaCADD on machines that do not have these capabilities built in.

To install DynaCADD:

1. With your computer off, insert the DynaCADD key into the joystick port and then turn your computer on.
2. Insert the DynaCADD 1 disk into any drive.
3. Double-click on the DynaCADD 1 icon.
4. Double-click on the InstallHD icon.
5. The InstallHD program will ask for the name of the destination hard drive or partition. After having entered a valid name, press RETURN or click on OK.

Loading DynaCADD - AMIGA Version

1. If your computer is turned off, make sure the DynaCADD program key is inserted. If it is not, shut your system off now and insert the key, then turn your system back on.
2. Locate the DynaCADD drawer on your hard disk and open it.
3. Click on the DynaCADD icon. This will run DynaCADD and bring you to the DynaCADD System level.

Installing DynaCADD - ST/TT Version

DynaCADD requires at minimum a ST or TT system with 1 Mb of RAM and a hard disk to operate properly. A math co-processor and 2 Mb of RAM is highly recommended. DynaCADD includes two specific versions:

1. ST with or without a math co-processor.
2. TT or ST with a 68030 accelerator card with a 68881 math co-processor.

Depending on your system configuration, choose the appropriate DynaCADD version during installation. The ST version of DynaCADD will automatically detect a math co-processor and take full advantage of it. A screen resolution of at least 640 by 400 is required.

To install DynaCADD:

1. With your computer off, insert the DynaCADD key into the joystick port and then turn your computer on.
2. Insert Disk 1 into your computer.
3. Double-click on the Drive icon.
4. Double-click on the HINSTALL.PRГ icon.
5. When asked source drive A or B, type the appropriate letter.
6. When asked which version you require, type "S" for ST, "T" for TT or "B" for both.
7. When asked which partition you require, type the appropriate letter.
8. When asked for a specific disk, insert the appropriate disk and press RETURN. The contents of each disk will now be copied to your hard disk.

Loading DynaCADD - ST/TT Version

To load DynaCADD:

1. Double click on (open) the disk drive icon where the Program is located.
2. If DynaCADD is installed in a folder, double click on (open) the folder within which the Program is located.
3. Double click on the DynaCADD program icon. This brings you to the System level.

Step 1: Establish the Drawing Parameters

Before starting this tutorial, DynaCADD must be properly installed into your system. If you have not already done so, please refer to "Installing DynaCADD" found within the first few pages of this Tutorial.

This step creates the appropriate files and establishes the basic drawing parameters.

1.A: Load the DynaCADD program. For instructions on how to load DynaCADD, refer to "Loading DynaCADD" found within the first few pages of this Tutorial.

1.B: The first time that you run DynaCADD, you enter the program at the System level. After that, DynaCADD keeps track of where you left off the last time a drawing was saved. When DynaCADD is run again, a dialog appears. If this is the case and the Auto Start dialog appears, select the "No" button to not continue with a previous session and enter the System level.

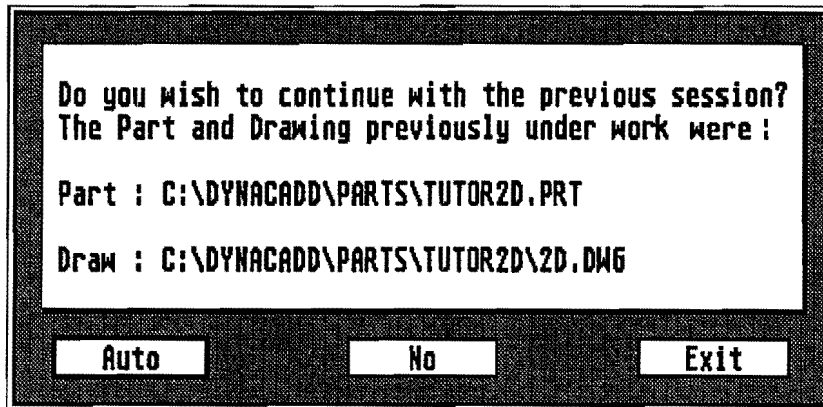


fig. (3) Auto Start Dialog

1.C: Activate/Create the Part

Description: Move the pointer over the “Activate Part” button and press the left mouse button. When the file selector appears, select the parts directory and type the name of a new part, “TUTORIAL” then press the [Return] key.

Result: This creates the part file. This is where the 2-D entities will be saved.

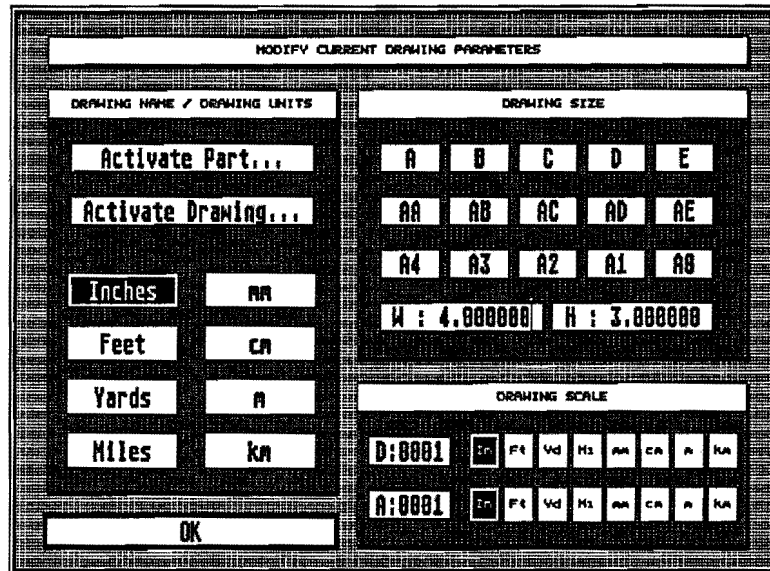


fig. (4) System Level Dialog

1.D: Activate Drawing

Description: Move the pointer over the “Activate Drawing” button and press the left mouse button. When the file selector appears, type the name of the new drawing, “2D” then press the [Return] key.

Result: This creates the drawing file associated with the part file.

Note: From now on, “Select” means move the pointer over the appropriate item and press then release the left mouse button.

1.E: Set the Drawing Unit

Description: Within the dialog, select the “Inch” button to select the drawing units to inches.

Result: This sets the basic units used in the drawing to be inches. Numbers will be entered and displayed in inches.

1.F: Set the Page Size

Description: Within the dialog, select the “A” button to select a “11.0” by “8.5” inch sheet.

Result: This sets the size of the drawing to be an A sized sheet. This limits the physical size of the paper to be 11 by 8.5 inches. However, since the drawing may be scaled, the entities in the drawing can represent any size in the “real” world.

1.G: Set the Drawing Scale

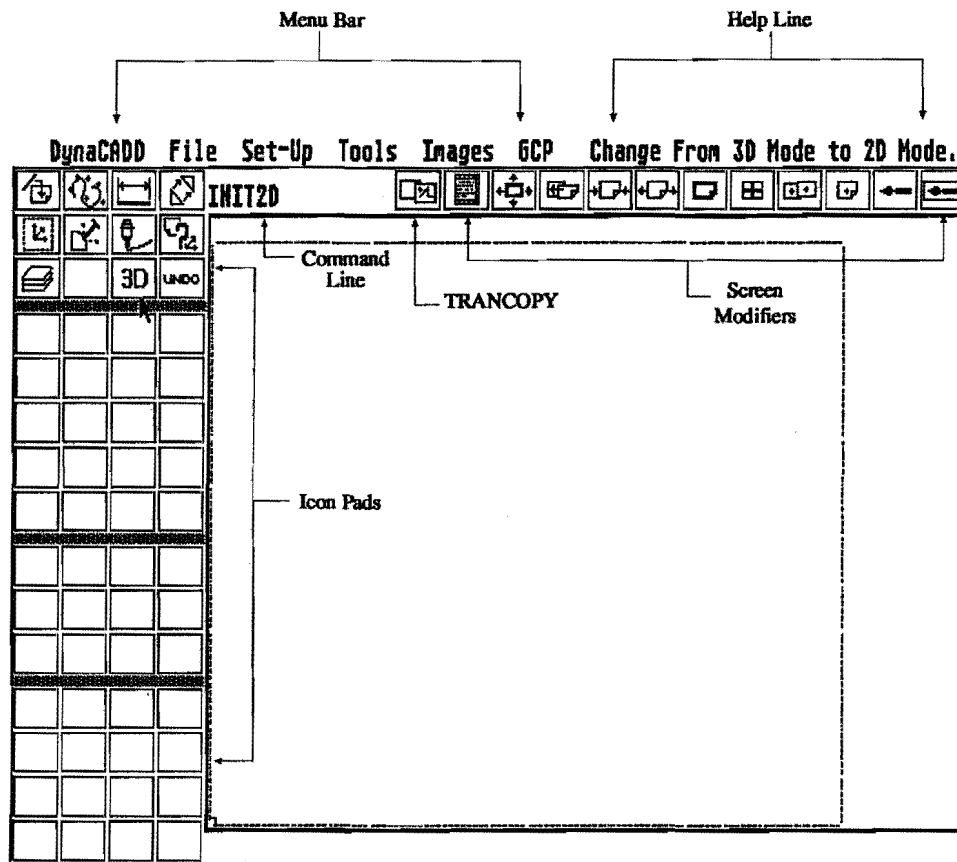
Description: Within the dialog, select the text box beside the “D”, type “1” and select inches. Select the text box beside the “A”, type “1” and select inches.

Result: This sets the scaling of the drawing to be 1.0. This means that one inch in the drawing equals one inch in the physical world.

1.H: Accept the Settings for the Drawing

Description: Select the button labeled “OK” in the dialog.

Result: This sets the drawing parameters to those just specified in the dialog. The dialog disappears and you now enter the CADD level.



Note: The position on the screen of the Help Line and the X-Y coordinates display may vary between platforms.

fig. (5) Entering the CADD Level

Enter the command sequence:



DZALL

Result: This zooms the entire drawing in the drawing area.

Step 2: Creating the First Entity

Select Pen Indexes from the Set menu.

In the dialog, select line weight 2 for a medium thick line. Move the pointer to the arrows beside the style names. Click the arrows to scroll the names until the line style CONTINUOUS appears. This is a solid line. Press [Return] to return to the CADD level.

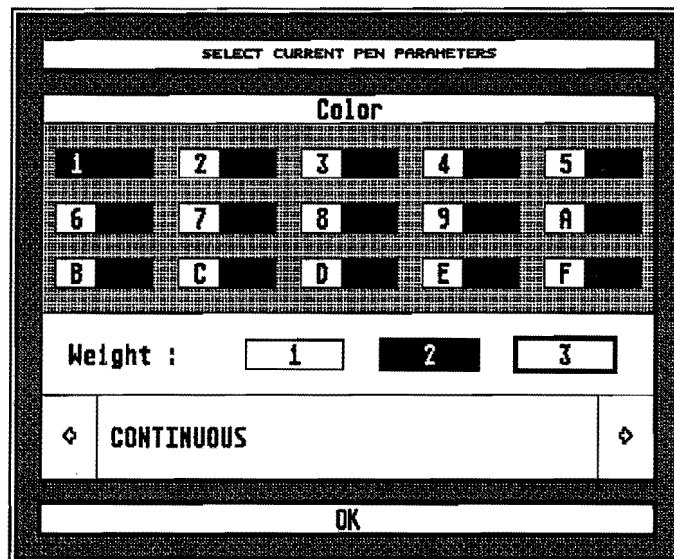


fig. (6) Set Menu, Pen Indexes

Enter the command sequence:



INSERT CIRCLE CIRRAD 0.15 [Return] [Return] .5,1.5 [Return] [;]

Move the pointer to the INSERT icon in the top left icon pad. The help line will show “INSERT”. Press and release the left mouse button to select the icon.

Move the pointer to the CIRCLE icon in the second from the top icon pad. The help line will show “CIRCLE”. Press and release the left mouse button to select the icon.

Move the pointer to the CIRRAD icon in the second from the bottom icon pad. The help line will show “CIRRAD”. When the CIRRAD command is selected, the calculator will appear so that it can accept the radius of the circle.

Enter “0.15”. This sets the radius of the circle to 0.15 inches.

Note: The calculator numbers and math operations can be entered using the key board or using the buttons on the calculator dialog.

Press the return key [Return] to end the calculator dialog and accept the value.

Press the return key [Return] a second time. This will bring up the absolute coordinates dialog.

Enter “.5,1.5”. This will anchor the center of the circle at .5,1.5.

Press the return key [Return]. This will end the absolute coordinates dialog, accepting the values entered.

Note: The “OK” button in the absolute coordinates dialog can be used instead of the [Return] key.

Press the semi-colon key [;] to end the command. The semi-colon key is equivalent to pressing the right mouse button.

Result: This inserts a circle, of radius 0.15 units, at location .5,1.5. This circle defines the circumference of the innermost well of the screw.

The process of creating a part involves inserting and modifying entities. An entity is the basic drawing primitive in DynaCADD and includes such objects as points, lines, circles and arcs. INSERT commands are described in Chapter 7.



fig. (7) Inserting a Circle

Step 3: Inserting a Line

Enter the command sequence:



INSERT LINE LINELOC ORG ent #1 LOC [I][Y] .5 [L];

Move the pointer to the INSERT icon in the top left icon pad. The help line will show "INSERT". Press and release the left mouse button to select the icon.

Move the pointer to the LINE icon in the second from the top icon pad. The help line will show "LINE". Press and release the left mouse button to select the icon.

Note: The LINELOC is the default command. It will automatically be started.

Move the pointer to the ORG icon in the bottom icon pad. The help line will show "ORG". Press and release the left mouse button to select the icon.

Note: An alternate method of selecting the "ORG" command is by pressing the [F2] function key.

Move the pointer to the outline of the circle. Press and release the left mouse button. This anchors the beginning of the line at the center of the circle.

Move the pointer to the LOC icon in the bottom icon pad. The help line will show "LOC". Press and release the left mouse button to select the icon.

Note: An alternate method of selecting the "LOC" command is through the [F1] function key.

Note: The LOC command must be active before any of the absolute or relative commands (X, Y, Z, IX, IY, IZ, PRAD, PANG) can be used.

Press the “I” and the “Y” keys ([I][Y]). This executes the IY command, and brings up the calculator dialog. The IY command increments from the previous location (in this case, the center of the circle) along the Y axis defining the new location.

Enter “.5”. This sets the length of the line segment along the Y axis to half an inch.

Note: The calculator numbers and math operations can be entered using the key board or using the buttons on the calculator dialog.

Press the comma key [,]. This closes the calculator dialog, accepting the value entered. The new line segment is drawn.

Press the semi-colon key [;] or the right mouse button to end the command.

Result: This creates a line segment from the center of the circle towards the top part of the drawing. The line is 0.5 of an inch in length.

Note: When selecting either absolute or relative coordinates using the calculator, the point can be anchored by closing the calculator with a comma [,]. Until the comma key ([,]) is pressed, the coordinates specified by one of the location commands (X, Y, Z, IX, IY, IZ, PRAD, PANG) will not be selected. To change the specific component of a location, such as the X component, simply close the calculator by pressing the [Return] key rather than the comma key ([,]).

Note: Some commands, like the INSERT LINE LINELOC, will continue until the right mouse button or the semi-colon key ([;]) is pressed. With INSERT LINE, the end of the first line will be the beginning of the next. By ending the command with a semi-colon or the right mouse button, the action is equivalent to “cutting” the continuous line at the current location.

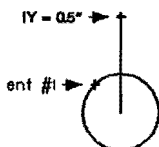


fig. (8) Inserting a Vertical Line

Step 4: Inserting a Horizontal Guide Line.

Enter the command sequence:



INSERT LINE HORIZONT ORG ent #1 LOC [I][X] 3.00 [.] [.]

Result: This inserts a long horizontal guide line from the center of the circle (the origin of entity #1) to the right side of the circle. This is a guide for locating the center of the screw. The length of the line is set for compatibility with future steps in this tutorial.

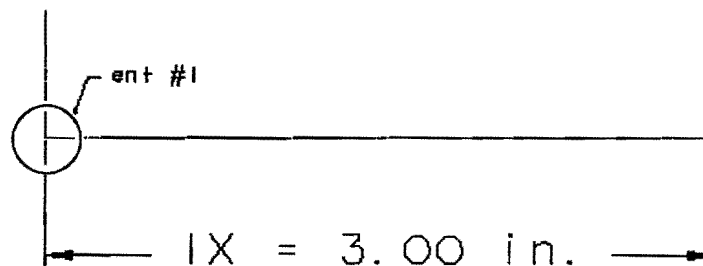


fig. (9) Inserting a Horizontal Guide Line

Step 5: Copy the Horizontal Line.

Enter the command sequence:



TRANSFORM TRANSLATE TRANCOPY ent #1 [:] END ent #1

INT ent #2 ent #3[:]

Result:

This copies the horizontal guide line from its original location and places the copy at the top of the circle. The original line is entity #1. The selected line is copied from the end of that line (entity #1) to the intersection of the circle (entity #2) and the vertical guideline (entity #3). This new line is a guide for the lower shank of the screw, and aligns with the inner circle (well).

TRANSFORM commands are described in Chapter 10.



fig. (10) Copying a Horizontal Line

Step 6: Change the Line Style

Enter the command sequence:



TRANSFORM MODIFY CHAPEN [line weight 1 and line style
STITCH] [Return] ent #1 [;]

Select line weight 1 and line style STITCH in the “Select Current Pen
Parameters” dialog.

Press [Return].

Result: This changes the current line style and weight of the last horizontal line. Note that this does not affect the current *pen* weight and style, it only changes the selected entities. This change is made so that the subsequent line does not appear as a center line.

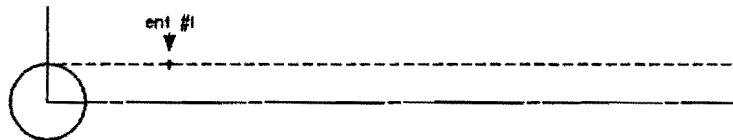


fig. (II) Changing the Line Style

Step 7: Insert a Horizontal Line

Select Pen Indexes from the Set menu.

Select line weight 2, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT LINE HORIZONT LINLEN 0.357 [Return] END ent #1 ent #2 [;]

Result: This draws a horizontal line 0.357 inches long from the end of entity #1, towards entity #2. This is the bottom segment of the shank.

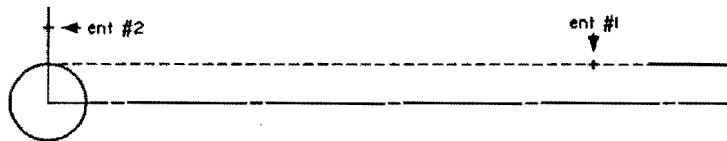


fig. (12) Inserting a Horizontal Line

Step 8: Hide the Line.

Enter the command sequence:



TRANSFORM MASK ent #1 [;]

Result: This hides the horizontal guide line. It still exists but cannot be modified until returned to view (unmasked). This is done because we can not snap to the necessary line if another line is attached to it at that location. Later, we will unmask the line, when we need to reference it.

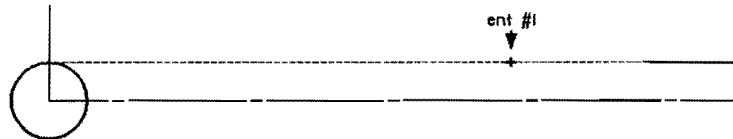


fig. (13) Hiding the Line

Step 9: Insert a New Line.

Enter the command sequence:



INSERT LINE LINELOC LINANG 120 [Return]

END ent #1 LOC loc #1 [;]

Result: This inserts another short line, at an angle of 120 degrees from the end of entity #1. Note that, in DynaCADD, the 0/360 degree mark is at the eastern position on the compass, and 90 degrees is at the north. This is the angled line that connects the lower and middle shanks.

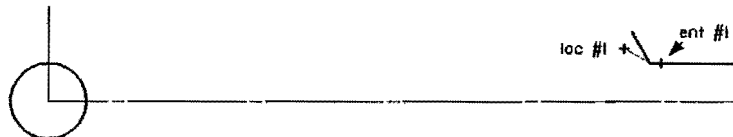


fig. (14) Inserting a New Line

Step 10: Insert Another Circle.

Enter the command sequence:



INSERT CIRCLE CIRDIA 0.374 [Return] ORG ent #1 [;]

Result: This draws a second circle with a diameter of 0.374 inches around the origin of the first (entity #1).
You could also use TRANSFORM OFFSET command to draw a second circle around the first. See Chapter 10.
This circle is the outer edge of the sunken well.

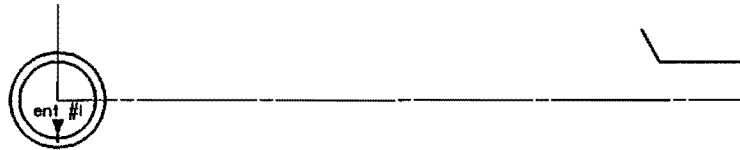


fig. (15) Inserting Another Circle

Step 11: Reveal Hidden Entities.

Enter the command sequence:



TRANSFORM UNMASK

Result: This reveals previously masked entities in the drawing.
We need to reveal the masked line because we are going to copy this line to create the middle shank.

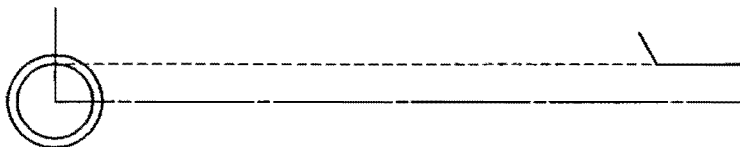


fig. (16) Revealing Hidden Entities

Step 12: Insert a Line Tangent to a Circle

Enter the command sequence:



TRANSFORM TRANSLATE TRANCOPY ent #1 [;] END ent #1
INT ent #2 ent #3 [;]

Result:

This copies the horizontal line (entity #1), to the intersection of the vertical line (entity #2) and the outer circle (entity #3). We need this line for the trim command, to follow.

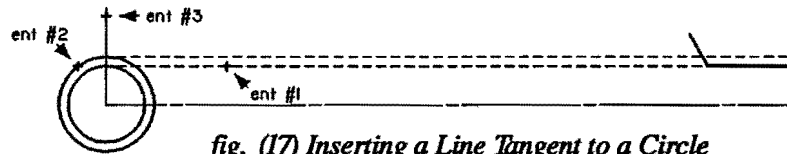


fig. (17) Inserting a Line Tangent to a Circle

Step 13: Trimming Entities

Enter the command sequence:



TRANSFORM TRIM TRIMMULTI ent #1 ent #2 [;] ORG ent #1 [;]

Result:

TRIM guides lines (entity #1 and entity #2) away from other entities. We needed the height of the trimmed horizontal line as a reference to determine the length of the angled line (connecting the shanks).

TRIM is a powerful feature that lets you lengthen or shorten an entity.

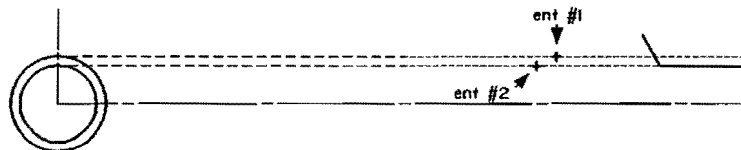


fig. (18) Trimming Entities

Step 14: Add More Horizontal Lines

Enter the command sequence:



INSERT LINE HORIZONTAL INT ent #1 ent #2 LOC loc #1 [;]

Result:

This draws a horizontal line from the point where entity #1 and entity #2 intersect. Since the lines do not physically intersect, DynaCADD calculates the point where they would intersect, if they were trimmed appropriately, and uses that point as the reference for the start of the new line.

The intersection of the guide line and the angled line is used as the reference for creating the middle shank.

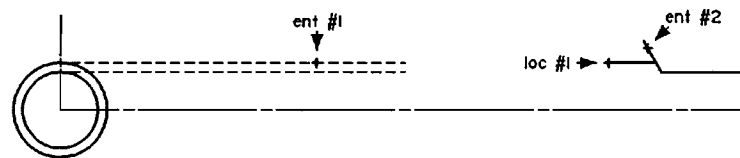


fig. (19) Adding Additional Horizontal Lines

Step 15: Trim Two Lines

Enter the command sequence:



TRANSFORM TRIM TRIMINTOF ent #1 ent #2 [;]

Result: This trims the angled line (entity #1) to the intersection of itself and entity #2. It also establishes the correct length of the connection between the two shank sections.

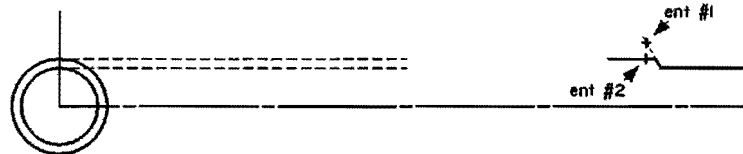


fig. (20) Trimming Two Lines

Step 16: Insert A Vertical Line

Select Pen Indexes from the Set menu.

Select line weight 1, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT LINE VERTICAL END ent #1 ent #2 [;]

Result: This draws a vertical line from the left end of entity #1 to the center line (entity #2). This is a guide line for creating the cutaway section marks.

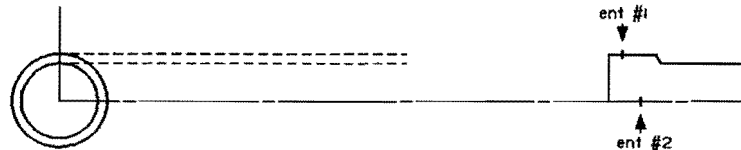


fig. (20) Inserting a Vertical Line

Step 17: Rotate a Vertical Line

Activate the TRANCOPY flag. This icon is found in the icon strip at the top of the drawing area.



When the TRANCOPY flag is selected, two images are displayed in the icon. Any transformation command applied to the selected entities will leave the original entities unchanged.



When the TRANCOPY flag is deselected, a single image is displayed in the icon. The selected entities will be deleted when a transformation command is applied to them.

Note: The only transformation commands not affected by this flag are the TRANSFORM TRANSLATE commands.

Enter the command sequence:



TRANSFORM ROTATE ROTANG 90 [Return] ent #1 [:] ORG ent #1 [:]

Result: This rotates a copy of the vertical line (entity #1) about its center by 90 degrees. This is the second guide line for creating the section lines.

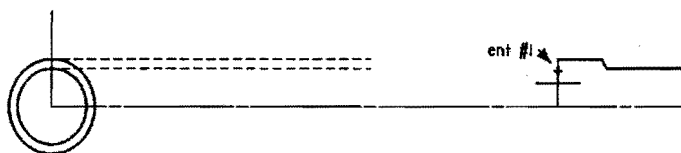


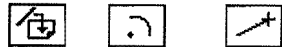
fig. (21) Rotating a Vertical Line

Step 18: Insert an Arc

Select Pen Indexes from the Set menu.

Select line weight 2, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT ARC END ent #1 ent #2 ent #3 [;]

Result:

This inserts an arc with its center at the end of the previously rotated line (entity #1) and its ends at the endpoints of the small vertical line (entity #2). This is the top portion of the cutaway area.

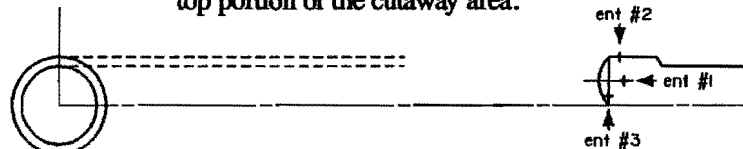


fig. (22) Inserting an Arc

Step 19: Mirror the Arc

Activate TRANCOPY, as in Step 17, so that the transformation command, TRANSFORM MIRROR, applied to the selected entities will leave the original entities unchanged.

Enter the command sequence:



TRANSFORM MIRROR ent #1 [;] END ent #2 ent #3 [;]

Result:

The arc is mirrored (copied across a defining line). This encloses the top part of the cutaway area.

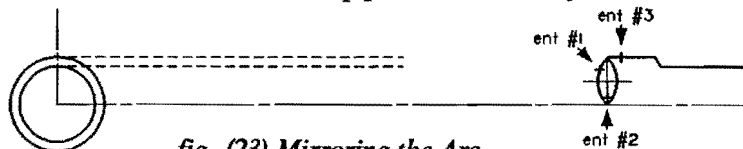


fig. (23) Mirroring the Arc

Step 20: Move the Vertical Line

Enter the command sequence:



TRANSFORM TRANSLATE TRANSMOVE ent #1 [;] ORG ent #2
END ent #2 [;]

Result:

This moves the small vertical line (entity #1) from the center to the left hand end of the small horizontal line (entity #2). This vertical line will be used as a mirror line in an upcoming step.

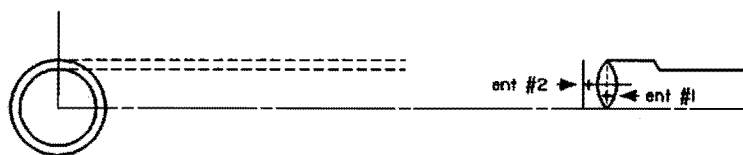


fig. (24) Moving the Vertical Line

Step 21: Delete Entities

Enter the command sequence:



TRANSFORM DELETE ent #1 [;]

Result:

This deletes the horizontal guide line, no longer necessary to our drawing. It is a good idea to periodically delete any guide lines which have no more use, in order to keep your drawing uncluttered.

The DELETE command in the INSERT icon pad is the same as this command (see Step 23).

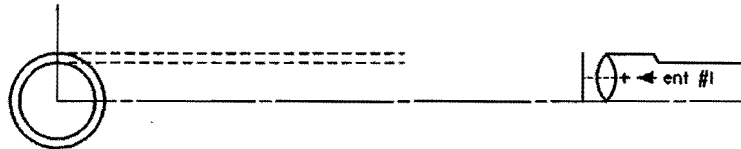


fig. (25) Deleting Entities

Step 22: Mirror Several Entities at Once

Activate TRANCOPY so that the transformation command, TRANSFORM MIRROR, applied to the selected entities will leave the original entities unchanged.

Enter the command sequence:



```
TRANSFORM MIRROR ent #1 ent #2 ent #3 [;]  
END ent #4 ORG ent #4 [;]
```

Result: This mirrors the horizontal line (entity #1) and the two arcs (entity #2, entity #3) across the vertical line determined by entity #4. This creates the middle shank and arcs for the other side of the cut away.

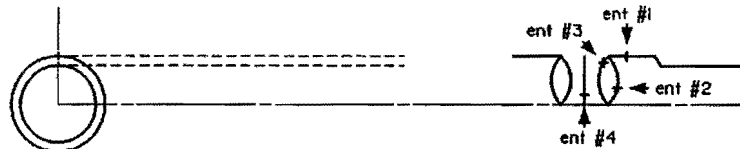


fig. (26) Mirroring Multiple Entities at Once

Step 23: Delete Entities

Enter the command sequence:



INSERT DELETE ent #1 [;]

Result: This deletes the unnecessary vertical guide line. It does not matter which delete command you select; either work equally well.

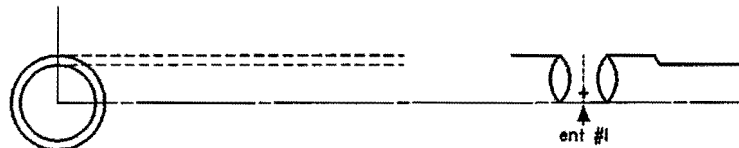
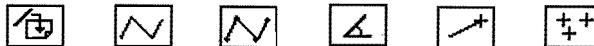


fig. (27) Deleting Entities

Step 24: Insert a Line

Enter the command sequence:



INSERT LINE LINELOC LINANG 160 [Return]
END ent #1 LOC loc #1 [;]

Result: This draws a line at an angle of 160 degrees from the end of entity #1. This is the angled connection between the middle and upper shanks of the screw.

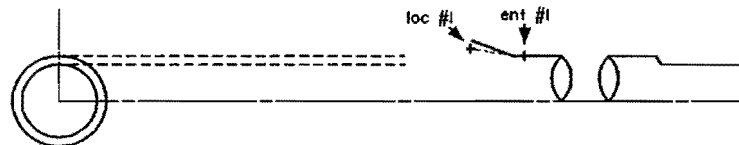


fig. (28) Inserting a Line

Step 25: Insert Orthographic Lines

Select Pen Indexes from the Set menu.

Select line weight 1, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT LINE ORTHO LINLEN 0.222 [Return] ON ent #1
LOC loc #1 loc #2 [;]

Result: This draws a guide line 0.222 inches from the center line (entity #1) and 0.222 inches from location #1 to location #2. These are used to determine the upper shank diameter.

Note: When selecting ON entity #1, choose a point closer to the left end of entity #1 (towards the circle) as opposed to the right end.

Ortho lines are either horizontal or vertical.

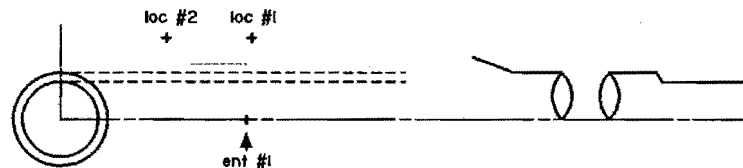


fig. (29) Inserting Orthographic Lines

Step 26: Insert a Horizontal Line

Select Pen Indexes from the Set menu.

Select line weight 2, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT LINE HORIZONT LINLEN 0.473 [Return] INT ent #1 ent #2
END ent #2 [;]

Result: This draws a horizontal line, length 0.473 inches, from the center line for the upper shank. The line is 0.222 inches from the center line, because you select the end of entity #2 and its exact size was entered in the previous step.

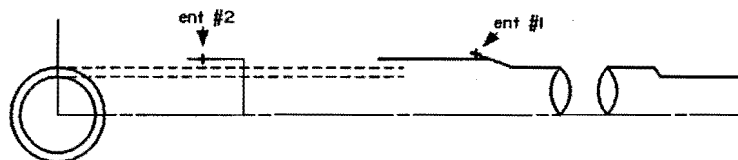


fig. (30) Inserting a Horizontal Line

Step 27: Trim the Angled Line

Enter the command sequence:



TRANSFORM TRIM TRIMONE ent #1 [:] INT ent #1 ent #2 [:]

Result: This trims the angled line to the intersection of the upper shank. This shows how trim can be used to match two lines to one another.

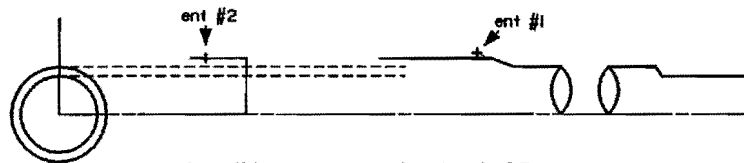


fig. (31) Trimming the Angled Line

Step 28: Divide a Line

Enter the command sequence:



TRANSFORM DIVIDE DIVLOC ent #1[:] SET END ent #1
LOC IX 0.18 [:] [:]

Result: This divides the line 0.18 inches along the X axis from the left endpoint, so that a smaller section of exact size can be moved later.

Divide breaks entities into smaller pieces except for curves (see Chapter 8). You cannot join entities together, only divide them.

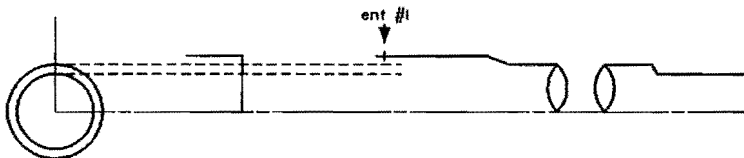


fig. (32) Dividing a Line

Step 29: Trim Several Lines

Enter the command sequence:



TRANSFORM TRIM TRIMMULTI ent #1 ent #2 ent #3 [;] END ent #4 [;]

Result: This trims the selected lines to the endpoint of a given line (entity #4). The guide lines and the center line are trimmed away from the front view of bolt, toward the right view.

This command permits you to trim multiple lines at once.

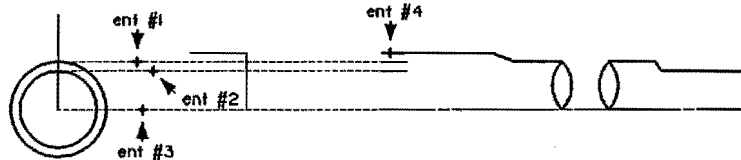


fig. (33) Trimming Multiple Lines

Step 30: Move a Horizontal Line

Enter the command sequence:



TRANSFORM TRANSLATE TRANSMOVE ent #1 [;] END ent #1
SET END ent #2 LOC IY 0.25 [;] [;]

Result: This moves the previously divided horizontal line a vertical distance of 0.25 inches above the center line. IY determines the relative distance along the Y axis.

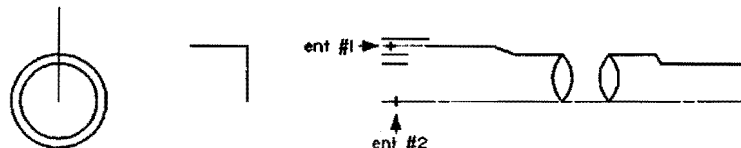


fig. (34) Moving a Horizontal Line

Step 31: Delete Entities

Enter the command sequence:



TRANSFORM DELETE ent #1 ent #2 ent #3 [;]

Result: Clean up the drawing area by deleting unnecessary guide lines.

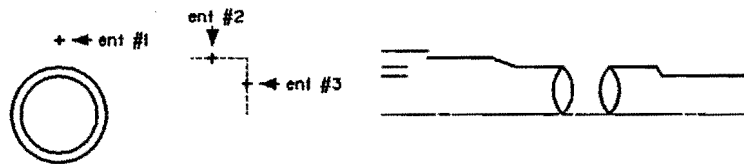


fig. (35) Deleting Entities

Step 32: Trim a Line to a Specified Distance

Enter the command sequence:



TRANSFORM TRIM TRIMDIST 0.222 [Return] ent #1 [;]

Result: This trims the horizontal line to exactly 0.222 inches. This makes sure the guide line matches the depth of the outer part of the sunken well. This command trims an entity to an exact length.

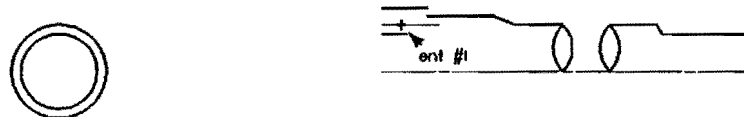


fig. (36) Trimming a Line

Step 33: Trim a Line to a Specified Distance

Enter the command sequence:



TRANSFORM TRIM TRIMDIST 0.36 [Return] ent #1 [;]

Result:

This trims the horizontal line to an absolute distance of 0.36 inches. The line is trimmed to the depth of inner part of the sunken well.

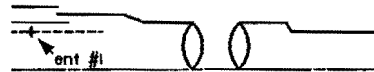
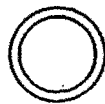


fig. (37) Trimming a Line

Step 34: Trim a Line to an Endpoint

Enter the command sequence:



TRANSFORM TRIM TRIMONE ent #1 [;] END ent #2 [;]

Result:

This trims the bottom line to the endpoint of the top line. The inner sunken well now matches the outer sunken well.

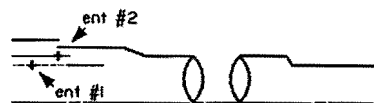
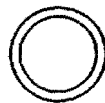


fig. (38) Trimming a Line to an Endpoint

Step 35: Change Pen Style

Enter the command sequence:



TRANSFORM MODIFY CHAPEN [Weight 1, Style HIDDEN] ent #1 ent #2 [:]

Result:

Change the pen weight and style of the selected entities to hidden lines. These entities would normally not be seen, so they are shown as hidden.

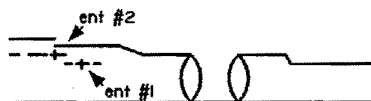


fig. (39) Changing the Pen Style

Step 36: Mirror the Right View

Activate TRANCOPY so that the transformation command, TRANSFORM MIRROR, applied to the selected entities will leave the original entities unchanged.

Enter the command sequence:



TRANSFORM MIRROR BYWIN loc #1 loc #2 [:] END ent #1 ORG ent #1 [:]

Result:

Mirror the right view across the axis to create the whole right view. This time a window is used to determine the entities to mirror.

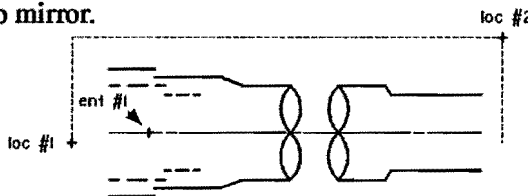


fig. (40) Mirroring the Front View

Step 37: Delete Arcs

Enter the command sequence:



TRANSFORM DELETE ent #1 ent #2 [;]

Result: This cleans up the unnecessary arcs away from the cutaway section.

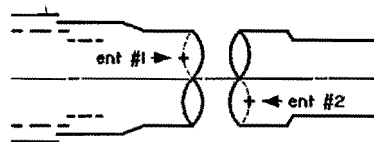


fig. (41) Deleting Arcs

Step 38: Insert Polygon

Enter the command sequence:



INSERT POLYGON POLYSIDES 6 [Return] POLYAFLAT 0.5 [Return]
ORG ent #1 END ent #1 [;]

Result: This inserts a hexagon (six sides) with a measurement of 0.5 inches across the flats. This polygon is used for the outside of the screw head. The reason END is chosen is to make the polygon start at zero degrees, thus aligning correctly with the view.

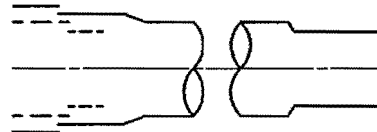


fig. (42) Inserting a Polygon

Step 39: Insert Section Lines

Select Pen Indexes from the Set menu.

Select line weight 1, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT SECTION SECTDIST 0.01 [Return] SECTANGLE 45 [Return]
ent #1 ent #2 ent #3 ent #4 [;]

Result: This inserts section lines between the selected arcs, 0.01 inches apart and placed at a 45 degree angle. These section lines are used in the cutaway view.

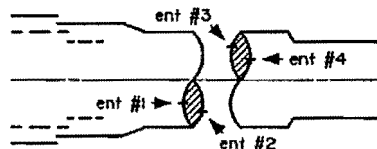


fig. (43) Inserting Section Lines

Step 40: Insert Vertical Lines

Select Pen Indexes from the Set menu.

Select line weight 1, line style HIDDEN and press [Return].

Enter the command sequence:



INSERT LINE VERTICAL END ent #1 ent #2 [:] ent #3 ent #4 [:]

Result: This inserts vertical lines to indicate the ends of the sunken well.

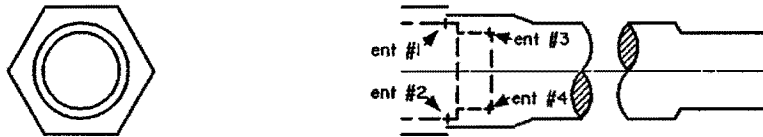


fig. (44) Inserting Vertical Lines

Step 41: Insert Vertical Lines

Select Pen Indexes from the Set menu.

Select line weight 2, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT LINE VERTICAL END ent #1 ent #2 [;] ent #3 ent #4 [;] ent #5
ent #6 [;] ent #7 ent #8 [;] ent #9 ent #10 [;] ent #11 ent
#12 [;] ent #13 ent #14 [;]

Result: This inserts vertical lines to define the various parts of the screw.

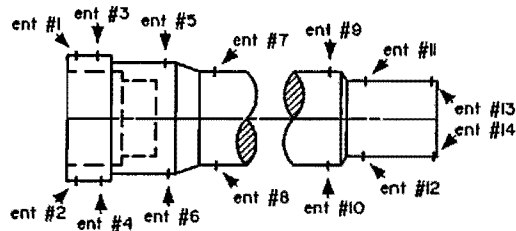


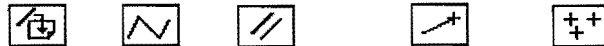
fig. (45) Inserting Vertical Lines

Step 42: Insert Parallel Line

Select Pen Indexes from the Set menu.

Select line weight 1, line style CONTINUOUS and press [Return].

Enter the command sequence:



INSERT LINE PARALLEL ent #1 END ent #2 LOC loc #1 [;]

Result: These reference lines are added to later correctly dimension the angles of the connections between shanks (see steps 54 and 55, below).

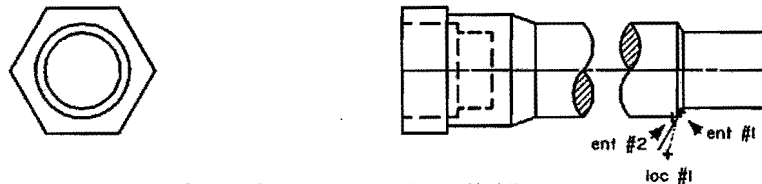


fig. (46) Inserting A Parallel Line

Step 43: Insert Parallel Line

Enter the command sequence:



INSERT LINE PARALLEL ent #1 END ent #2 LOC loc #1 [;]

Result: This inserts angled reference lines for dimensioning (see Step 42, above).

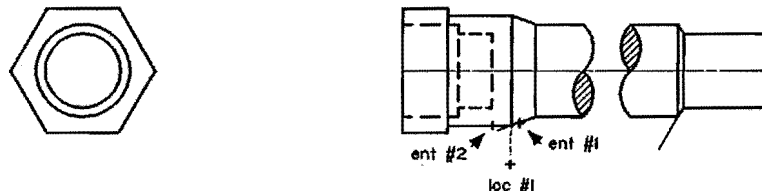


fig. (47) Inserting a Parallel Line

Dimensioning

You have now completed the basic drawing of the part. It should look like this:

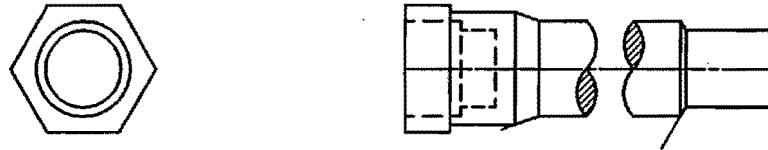


fig. (48) Completed Adjusted Screw Drawing without Dimensioning

The subsequent steps are all concerned with inserting dimensions into the drawing.

Dimensions provide measurements of specific details. DynaCADD automatically displays the specified distance (linear dimension) or angle (angular dimension) in the correct units (in this case, inches) or degrees, respectively.

Most technical drawings require accurate dimensions for every detail, in order to reproduce the part correctly. To replicate this process, the tutorial shows which type of dimension is required for each measurement.

Dimensions are one of the most important features of DynaCADD, and the commands are described in detail in Chapter 9. It isn't necessary to add every dimension described below, except to complete the drawing. However, it's a good opportunity to learn how to insert dimensions and to experiment with them.

There are many different types of dimensions, all of which cannot be covered here.

Step 44: Dimension Parameters

Select Dimension from the Set menu.

Select from the ARROWHEAD TYPE area: ARROWHEAD, ARROWS IN and ARROWS FILL.

Select from the DIMENSION TEXT FORMAT area: DECIMAL, IN LINE and UNIDIRECT.

Select from the AUTO TOLERANCING area: TOL. OFF.

Select as Precision: 3.

DEFINE DIMENSION PARAMETERS

ARROWHEAD TYPE


 ARROWHEAD


 CIRCLE


 SLASH


 ARROWS IN


 ARROWS OUT


 ARROW FILL


 ARROW OPEN

DIMENSION TEXT FORMAT


 ARCH


 DECIMAL


 IN LINE


 ABOVE


 BELOW


 UNIDIRECT


 ANGLED


 ALIGN

SUPPRESS

AUTO TOLERANCING


 LIMITS


 VARIATION1


 VARIATION2

PRECISION

fig. (49) Dimension Parameters Dialog

DynaCADD® 2-D Tutorial

Select SET GLOBALS and select the VERIFY TEXT option. Click OK. This forces DynaCADD to display the dimension text for verification before it is inserted. The text can then be modified in the dialog box.

When you return to the main dimension parameter dialog box, click OK.

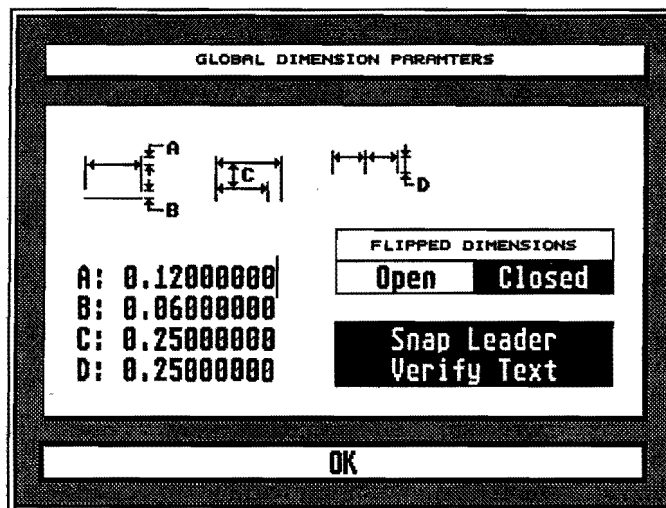


fig. (50) Global Dimension Parameter Dialog

Enter the command sequence:



CREATE LOADFONT LEROY__R.FNT

Load the font LEROY__R.FNT from disk. You must have a font loaded in order to dimension.

Enter the command sequence:



DIMENSION DIMHORI TEXTWID 0.075 [Return] TEXTHEI 0.075 x
1.25 [Return] END ent #1 ent #2 LOC loc #1 [;]
(Replace the number with 0.578)

Result:

This sets up the default dimension parameters in the dialog box and the width and height of the dimension text. The first dimension is inserted where indicated.

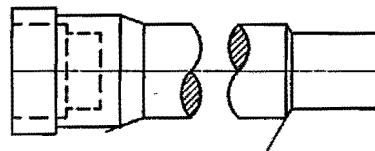
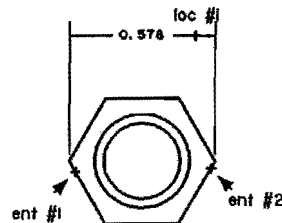


fig. (51) Dimension Width of Polygon

Step 45: Place Center Lines in a Circle

Enter the command sequence:



DIMENSION DIMCLINE ent #1

Result:

This inserts a center line on the selected circle.
DIMCLINE requires only that you select the correct type
of entity.

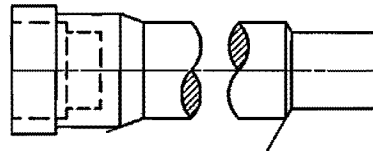
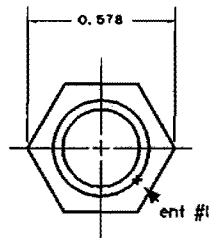


fig. (52) Placing Center Lines in a Circle

Step 46: Dimension Radius of an Inner Circle

Enter the command sequence:



DIMENSION DIMRAD ent #1 loc #1 [;]

Result: This dimensions the radius of the outer hidden circle. The location tells DynaCADD where to place the dimension line and text, relative to the origin of the selected entity.

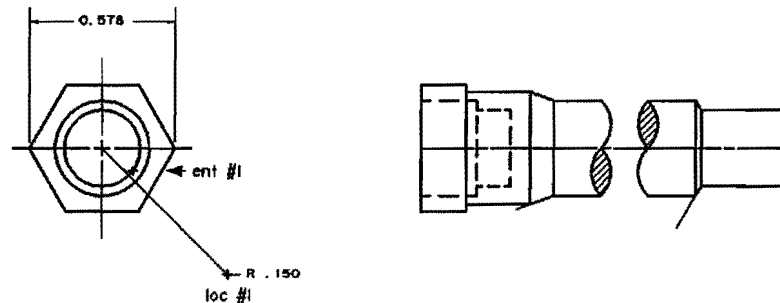


fig. (53) Dimensioning the Radius of an Inner Circle

Step 47: Dimension Diameter of an Outer Circle

Enter the command sequence:



DIMENSION DIMDIA ent #1 loc #1 [;]

Result: This dimensions the diameter of the inner hidden circle. This is similar to DIMRAD, in Step 46, except that it displays the diameter, rather than the radius.

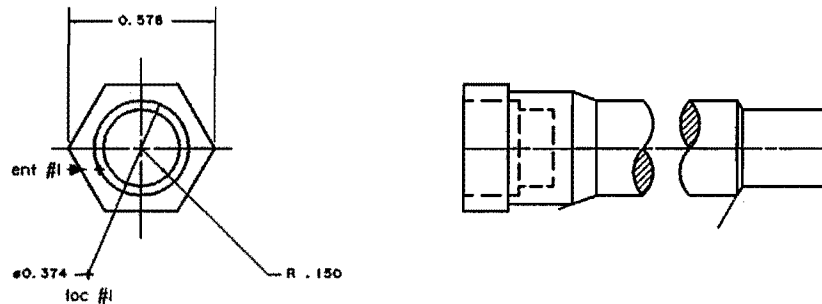


fig. (54) Dimensioning the Diameter of an Outer Circle

Step 48: Dimension the Hidden Section

Enter the command sequence:



DIMENSION DIMHBASE END ent #1 INT ent #2 ent #3 INT ent #3
ent #4 LOC loc #1 [;]

Result: This inserts the dimension of the hidden section, using horizontal baseline dimensioning. This inserts dimensions for successive distances (location 1, 2 and 3). The last selected location (#1) determines how far away to place the dimension text.

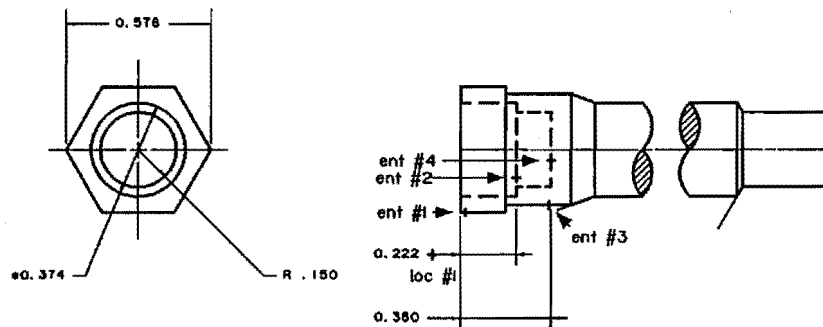


fig. (55) Dimensioning the Hidden Section

Step 49: Dimension Line Segments

Enter the command sequence:



DIMENSION DIMLINE ent #1 loc #1 [:] ent #2 loc #2 [:] ent #3 loc #3 [:]

Result: This inserts dimensions for the selected line segments.
The location determines how far away to place the dimension text.

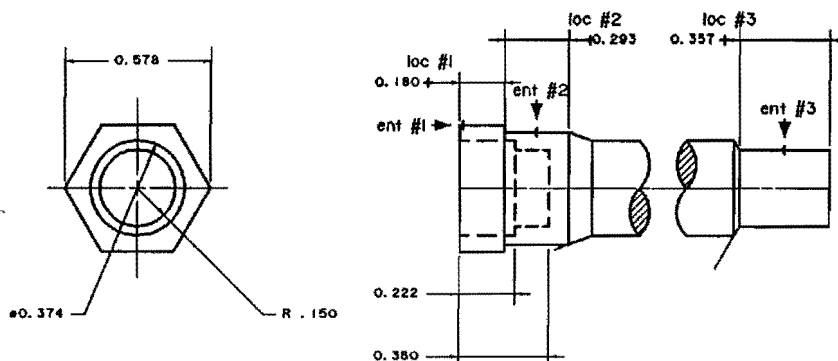


fig. (56) Dimensioning Line Segments

Step 50: Dimension Length of the Part

Enter the command sequence:



DIMENSION DIMHORI END ent #1 ent #2 LOC loc #1
[Replace the number with 3.000]

Result: This dimensions the true length of the screw by measuring from the ends of the entities.

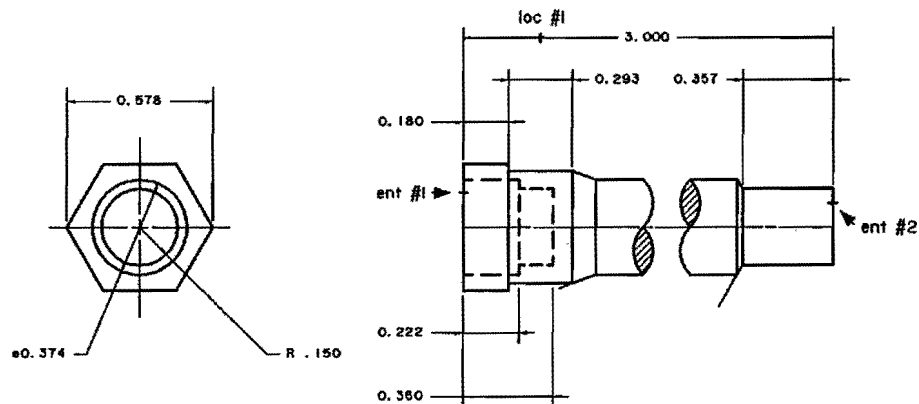


fig. (57) Dimensioning the Length of the Part

Step 51: Dimension Radius of the Upper Section

Select Dimension from the Set menu.

Select the following from the DIMENSION TEXT FORMAT area:
ANGLED, ABB'R and LEAD ZERO (S).

Enter the command sequence:



DIMENSION DIMVERT ORG ent #1 END ent #1 LOC loc #1 [;]

When the text display appears, move the cursor with the arrow keys before the dimension value. Type: R [space] and press [Return]. This indicates a radius, although the actual command only dimensions a vertical distance.

Result: This changes the initial dimension parameters as indicated. Then it inserts the dimension for the first section, or shank.

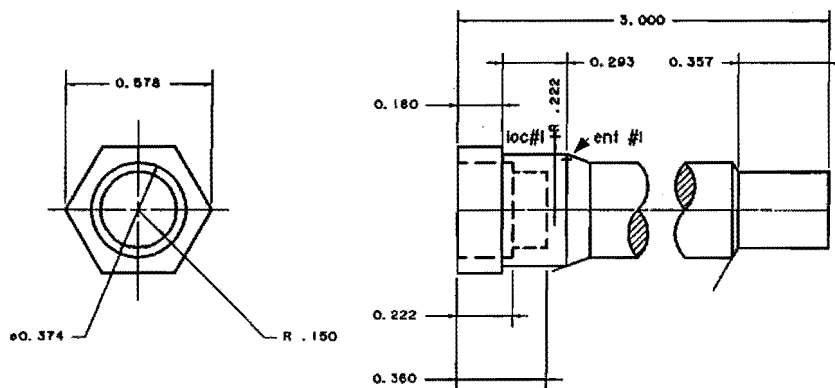


fig. (58) Dimensioning the Radius of the Upper Section

Step 52: Dimension Radius of the Middle Section

Enter the command sequence:



DIMENSION DIMVERT ORG ent #1 END ent #1 LOC loc #1 [;]

When the text display appears, move the cursor with the arrow keys before the dimension value. Type: R [space] and press [Return]. This indicates a radius, although the actual command only dimensions a vertical distance.

Result: This inserts the dimension of the middle section or shank.

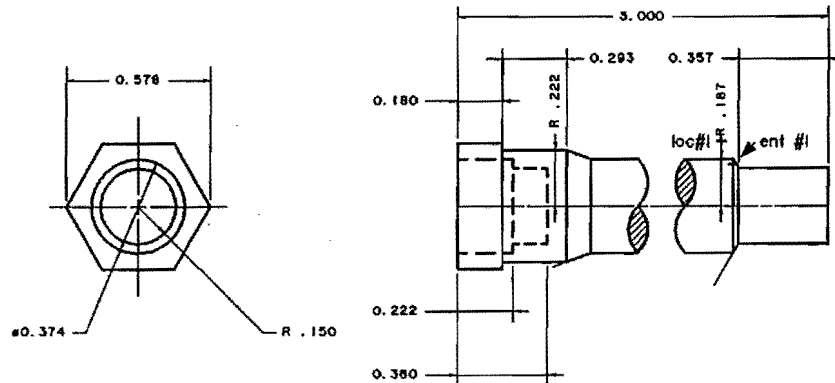


fig. (59) Dimensioning the Radius of the Middle Section

Step 53: Dimension Radius of the Lower Section

Enter the command sequence:



DIMENSION DIMVERT ORG ent #1 END ent #1 LOC loc #1

When the text display appears, move the cursor with the arrow keys before the dimension value. Type: R [space] and press [Return]. This indicates a radius, although the actual command only dimensions a vertical distance.

Result: This inserts the dimension for the end section or shank.

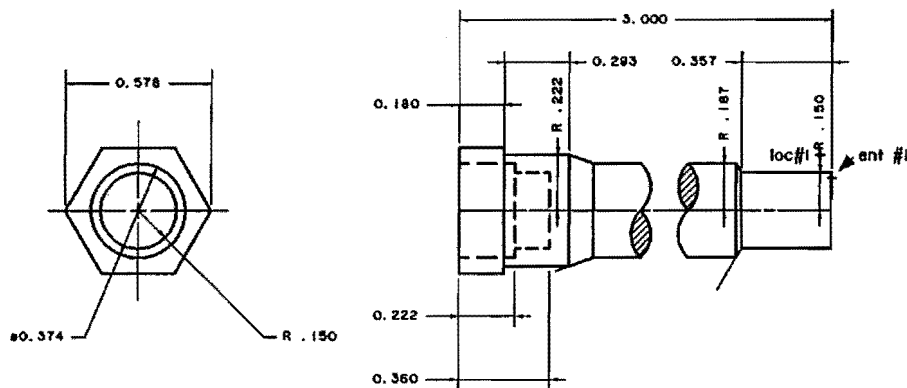


fig. (60) Dimensioning the Radius of the Lower Section

Step 54: Slope between Lower and Middle Sections

Select Dimension from the Set menu.

Select from the DIMENSION TEXT FORMAT area: UNIDIRECT, and DEG.

Select 0 places as the precision.

Enter the command sequence:



DIMENSION DIMANGLE ent #1 ent #2 END ent #2 [;]

Result: This inserts the dimension of the first slope, between the upper and middle shanks.

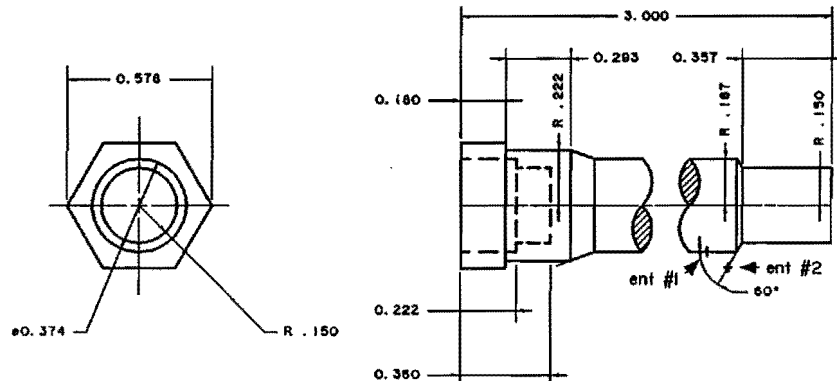


fig. (61) Inserting the Dimension of a Slope

Step 55: Slope between Middle and Upper Section

Enter the command sequence:



DIMENSION DIMANGLE ent #1 ent #2 END ent #2 [;]

Result: This dimensions the angle of the second slope, between the middle and lower shanks.

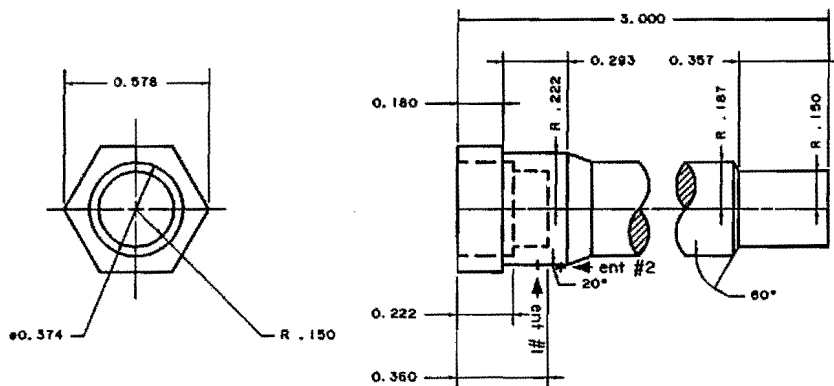


fig. (62) Inserting the Dimension of a Slope

Step 56: Create Gap on Sloped Lines

Enter the command sequence:



TRANSFORM TRIM TRIMINC -0.02 [Return] ent #1 [;] ent #2 [;]

Result: This trims the extended lines away from the main body of the part.

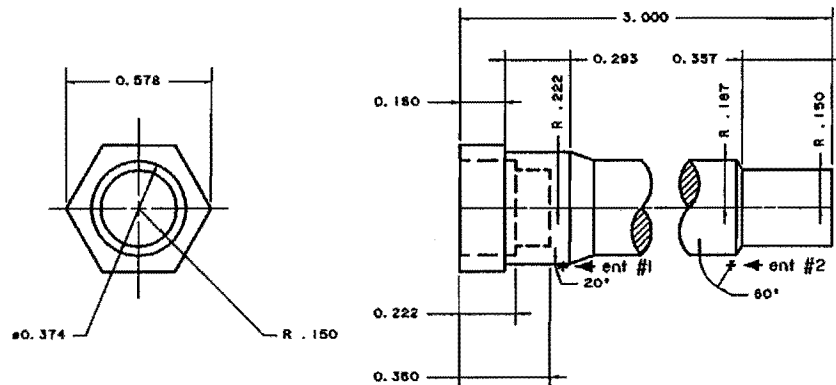


fig. (63) Creating a Gap on Sloped Lines

This completes our 2-D tutorial. Although only a few of the many DynaCADD commands were covered, it should give you the confidence to go on and try other 2-D commands.

Spend some time experimenting and exploring the 2-D commands before attempting to do the 3-D tutorial. The vast majority of CADD applications are in 2-D and this area should be mastered before you continue. In particular, examine how the different location and entity selection options affect the results of a command.

You should also try to print or plot your drawing. Chapter 13 discusses output in detail.

Now that you have created your first part and drawing, you should also read Chapters 1 through 5, if you have not already done so. These give you an overview of the program, the basic theory and practice of CADD, as well as providing specific information about DynaCADD itself.

Chapters 6 through 15 describe the DynaCADD menus and commands in detail. You should browse through this material in order to get a general idea of how the program is organized and what type of features and functions are available.

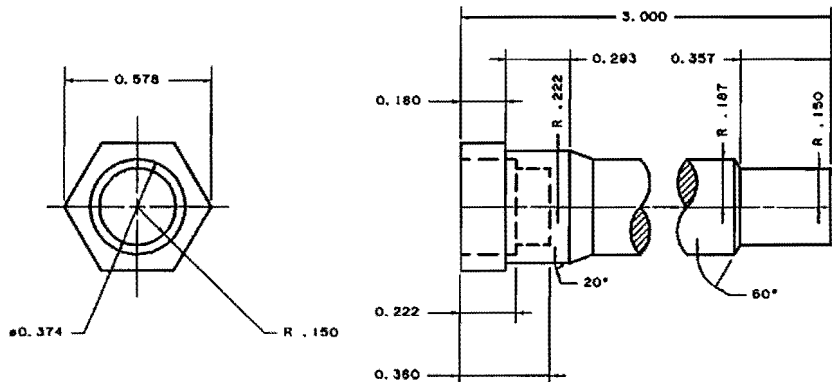


fig. (64) Completed and Dimensioned Adjusting Screw Drawing

DynaCADD 3-D Tutorial

This tutorial shows you how to construct a simple 3-D part with DynaCADD.

3-D mode has several differences from 2-D mode. 3-D mode has 3 dimensions, X, Y and Z, which correspond to width, height and depth, while 2-D mode has only width and height. As well, many of the DynaCADD commands are slightly different, and some additional commands are available.

One basic dilemma with 3-D is representing an object with 3 dimensions on 2 dimensional media (the screen or printer). This is handled by "viewing" the object from one direction. More than one view can be used at a time, but each view only shows the object from one direction. For example, when you look at a coffee cup from the top, you see a different "view" than when you look at the cup from the side.

Another dilemma is dealing with details that are "inside" the object, or behind other details in a particular view. DynaCADD shows these inside details as well as the overlaying ones.

For example, a drawing of a coffee cup will have all features visible, including the far side which would be hidden for a real cup.

Introduction

This tutorial creates a 3-D drawing of a caster wheel assembly. There are 4 elements to this assembly, and the finished assembly is shown in fig. (1). Although the figure may look daunting at first, this tutorial will demonstrate how DynaCADD can be used to create this drawing in 27 easy steps.

Disclaimer:

Since this is a tutorial intended to teach the use of DynaCADD, it does not create the drawing to industry standards. For example,

DynaCADD® 3-D Tutorial

this tutorial includes the added stage of drawing faces. However, using DynaCADD to create industry standard drawings is no more difficult than this tutorial.

Note: This tutorial has been optimized to show many “tricks” of the trade as well as demonstrating the basic 3-D part creation. Thus, it may be advantageous to review this tutorial after completing a few of your own drawings.

Note: Many of the commands in this tutorial use absolute numbers for commands which could be entered free hand using the mouse and pointer (for example, the zoom commands). However, the use of absolute numbers means that the display will consistently follow the tutorial.

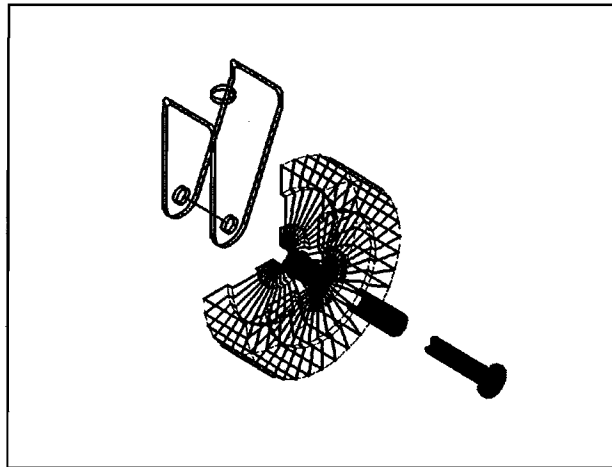


fig. (1) Completed 3-D Caster Assembly

Step 1: Establish the Drawing Parameters

Before starting this tutorial, DynaCADD must be properly installed into your system. If you have not already done so, please refer to "Installing DynaCADD" found within the first few pages of the DynaCADD 2-D Tutorial.

This step creates the appropriate files and establishes the basic drawing parameters.

1.A: Load the DynaCADD program. For instructions on how to load DynaCADD, refer to "Loading DynaCADD" found within the first few pages of the DynaCADD 2-D Tutorial.

1.B: The first time that you run DynaCADD, you enter the program at the System level. After that, DynaCADD keeps track of where you left off the last time a drawing was saved. When DynaCADD is run again, a dialog appears. If this is the case and the Auto Start dialog appears, select the "No" button to not continue with a previous session and enter the System level.

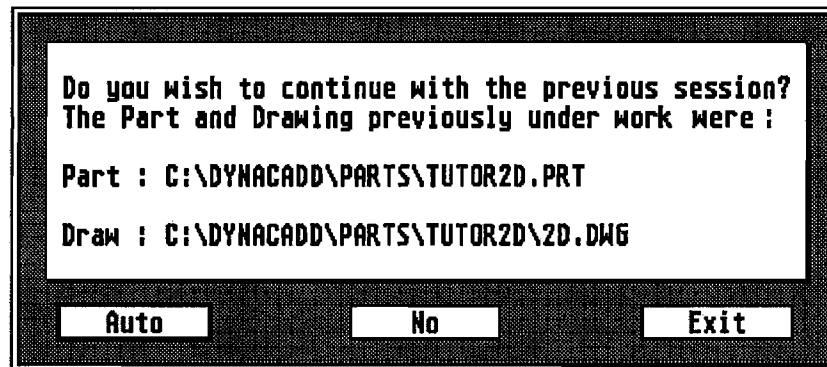


fig (2) Auto Start Dialog

1.C: Activate/Create the Part

Description: Move the pointer over the “Activate Part” button and press the left mouse button. When the file selector appears, select the parts directory and type the name of a new part, “CASTER” then press the [Return] key.

Result: This creates the part file. This is where the 3-D entities will be saved.

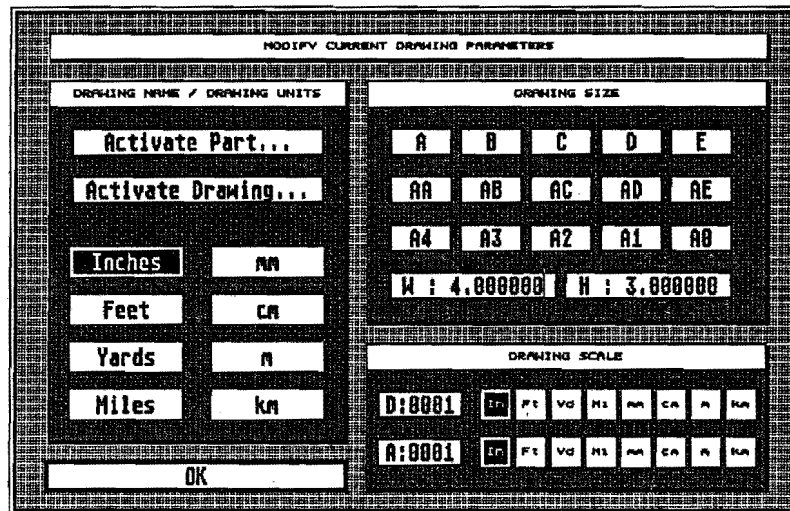


fig. (3) System Level Dialog

1.D: Activate Drawing

Description: Move the pointer over the “Activate Drawing” button and press the left mouse button. When the file selector appears, type the name of the new drawing, “WHEEL” then press the [Return] key.

Result: This creates the drawing file associated with the part file.

Note: From now on, “Select” means move the pointer over the appropriate item and press then release the left mouse button.

1.E: Set the Drawing Unit

Description: Within the dialog, select the “Inch” button to select the drawing units to inches.

Result: This sets the basic units used in the drawing to be inches. Numbers will be entered and displayed in inches.

1.F: Set the Page Size

Description: Within the dialog, select the “A” button to select a “11.0” by “8.5” inch sheet.

Result: This sets the size of the drawing to be an A sized sheet. This limits the physical size of the paper to be 11 by 8.5 inches. However, since the drawing may be scaled, the entities in the drawing can represent any size in the “real” world.

1.G: Set the Drawing Scale

Description: Within the dialog, select the text box beside the “D”, type “1” and select inches. Select the text box beside the “A”, type “1” and select inches.

Result: This sets the scaling of the drawing to be 1.0. This means that one inch in the drawing equals one inch in the physical world.

Note: Each 3-D view has an additional scaling factor for that view.

1.H: Accept the Settings for the Drawing

Description: Select the button labeled “OK” in the dialog.

Result: This sets the drawing parameters to those just specified in the dialog. The dialog disappears and you now enter the CADD level.

Step 2: Create a View

2.A: Zoom the Drawing

Enter the command sequence:



DZALL

Description: Move the pointer to the DZALL command in the top right icon pad running along the top of the display. The help line will show "DZALL". Press then release the left mouse button to select the icon.

Result: This zooms the drawing so that only the drawing page is visible in the drawing area.

Note: An alternative method of selecting the "DZALL" command is through the [F8] function key.

2.B: Change to 3D Mode

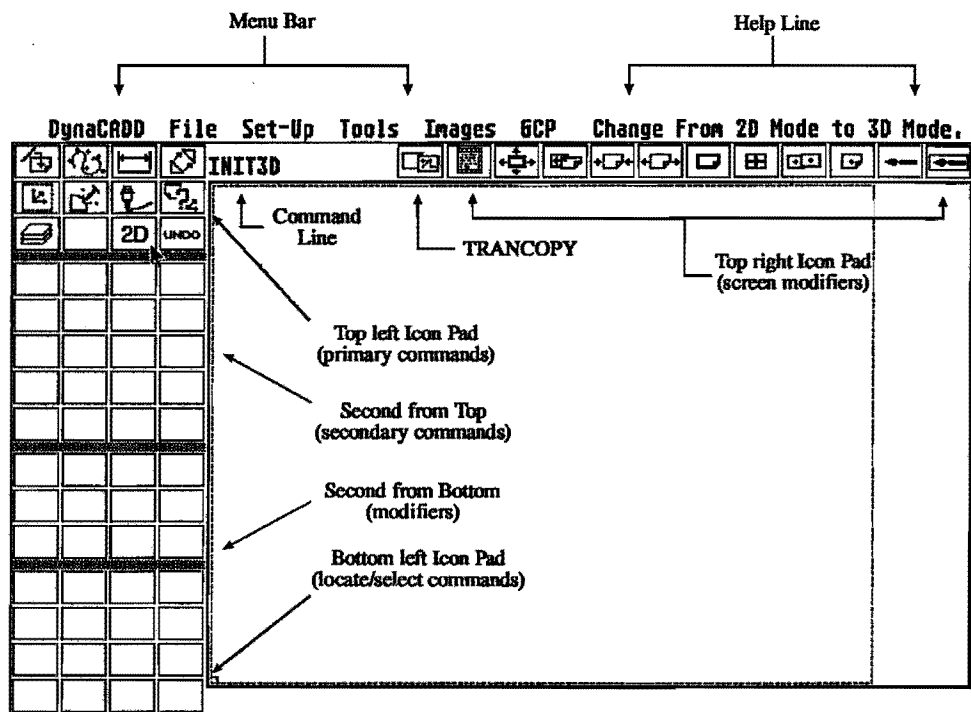
Enter the command sequence:



INIT3D

Description: Move the pointer to the INIT3D icon in the top left icon pad. The help line will show "INIT3D". Press and release the left mouse button to select the icon.

Result: This sets DynaCADD into 3-D mode. All operations will now be done in 3-D mode. Refer to fig. (4).



Note: The position on the screen of the Help Line and the X-Y coordinates display may vary between platforms.

fig. (4) Entering the CADD Level

2.C: Select Current GCP as Top

Enter the command sequence:

Menu "GCP", select "Top"

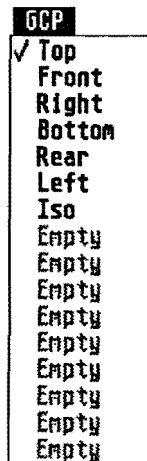
Description: Select the item labeled "Top" from the menu labeled "GCP".

Result: This sets the current GCP to Top. Any drawing modifications will be performed looking at the drawing from the top. Thus the view will be created so that the view looks at the part from the "Top".

Note: The actual GCP of the view may be different, but the modification will be performed in the current GCP.

Note: The current GCP is indicated in the GCP menu by a check mark beside the appropriate menu item.

Note: The current GCP may already be set to Top. Reselecting the already current GCP has no effect.



*fig. (5) GCP Menu,
Top Selected*

2.D: Create the View

Enter the command sequence:



EDITVIEW DEFVIEW [Return] 3,5 [Return] [;]

Description: Move the pointer to the EDITVIEW icon in the top left icon pad. The help line at the top middle left of the display will show the command name “EDITVIEW” as the pointer moves over the icon. Press and release the left mouse button to select the icon.

Move the pointer to the DEFVIEW icon in the second from the top icon pad. The help line will show “DEFVIEW”. Press and release the left mouse button to select the icon.

Press (and release) the return key ([Return]). This will bring up the absolute coordinates dialog as shown in fig. (6).

Enter “3,5” (without the quotes). This sets the origin of the view to a convenient position.

Press (and release) the return key ([Return]). This will end the absolute coordinates dialog, accepting the values entered in the last step.

Press the right mouse button, OR press the semi-colon [;] key. This defines the bounds of the view to the full extents of the physical page. (The view fills the entire drawing page.)

Input Absolute [X], [Y], [Z] Co-ordinates _____	OK
--	---------------

fig. (6) Absolute Coordinates Dialog

Result: This creates a 3-D view with the view origin at 3,5 and the area of the view at the full drawing page. fig. (7) shows the resulting display.

Note: The GCP indicator will become visible. It is the outlined double arrow at the center of the drawing.

Note: The GCP of the view will be set to Top (the current GCP). The scale of the view is the default value of 1.0.

Aside: Many of modifiers associated to the commands are initially set to a value. This value is known as a default value. If the modifier is selected, it may retain the value it is set to. Other modifiers, however, will always retain the default value unless otherwise set.

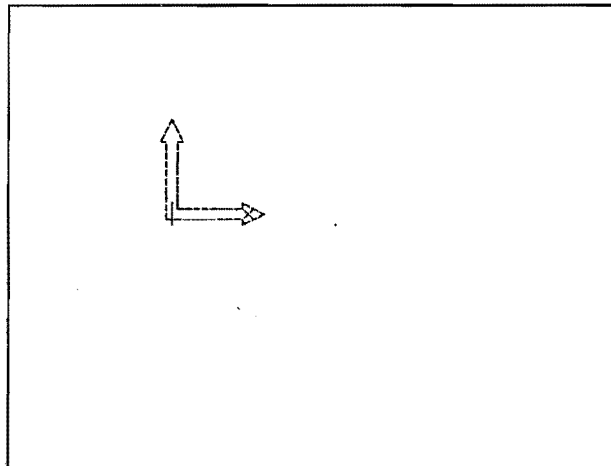


fig (7) 3-D View Created

Step 3 : Draw the Bottom Lines of the First Piece

Enter the command sequence:



INSERT LINE LINELOC [Return] 0,0,0 [Return] [I] [X] 19/32 [,] [I] [X]
5/16 [,] [Return] i2 [Return] [;]

Description: Move the pointer to the INSERT icon in the top left icon pad. The help line will show “INSERT”. Press and release the left mouse button to select the icon.

Move the pointer to the LINE icon in the second from the top icon pad. The help line will show “LINE”. Press and release the left mouse button to select the icon.

Note: The LINELOC is the default command. It will be automatically started.

Press the return key ([Return]). This will bring up the absolute coordinates dialog.

Enter “0,0,0”. This will anchor the start of the line to the origin at 0,0,0.

Press the return key ([Return]). This will end the absolute coordinates dialog, accepting the values entered in the last step.

Note: The “OK” button in the absolute coordinates dialog can be used instead of the [Return] key.

Press the “I” and the “X” keys ([I] [X]). This executes the IX command, and brings up the calculator dialog shown in fig. (8). The IX command increments from the previous location (in this case 0,0,0) along the X axis, defining the new location.



Aside: The IX command can also be selected using the IX icon in the bottom icon pad.

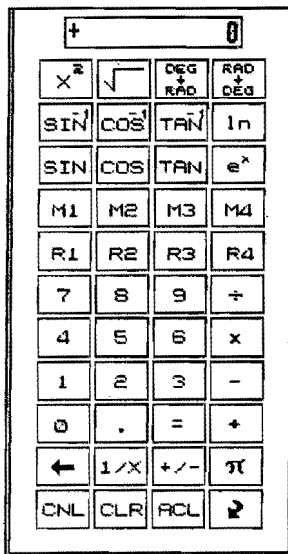


fig. (8) Value Calculator

Enter "19/32". This sets the length of the line segment along the X axis to 19/32 of an inch.

Aside: The calculator numbers and math operations can be entered using the key board or using the buttons on the calculator dialog.

Press the comma key ([,]). This closes the calculator dialog, accepting the value entered. The new line segment is drawn. The INSERT LINE LINELOC command is ready for a new line segment, starting at the end of the previous line.

Note: Closing the calculator dialog with a comma ([,]) indicates that the point being entered is fully specified.

Closing the calculator using the [Return] key indicates that the point is not fully specified yet.

Press the "I" and the "X" keys ([I] [X]). This executes the IX command, and brings up the calculator dialog again.

Enter "5/16". This sets the length of the line segment along the X direction to 5/16 of an inch.

Press the comma key ([,]). This closes the calculator dialog, accepting the value entered. The new line segment is drawn from the end of the previous line.

The INSERT LINE LINELOC command is again ready for a new line segment, starting at the end of this last line.

Aside: This line is drawn in three segments (one still to be entered) for reasons which will become apparent in future steps.

Press the return key ([Return]). This will bring up the absolute coordinates dialog.

Enter "i2". This specifies a location 2 inches further along in the X direction from the immediately preceding location (end of the last line).

Note: The "i2" means add 2 to the previous X coordinate to get the new location X value.

Aside: The IX command could have been used here with the same effect as "i2".

Press the return key ([Return]). This will end the absolute coordinates dialog, accepting the values entered in the last step. This enters a third line segment, from the end of the previous segment, two inches long.

Result:

This creates 3 line segments, all in the same line. The first line segment is 19/32 inches long, the second is 5/16 inches long, and the third is 2 inches long.

Note: Since the endpoints of the line segments are connected, the three lines will appear as a single line.

Note: When selecting either absolute or relative coordinates using the calculator, the point can be anchored by closing the calculator with a comma [,]. Until a comma key ([,]) is pressed, the coordinates specified by one of the location commands (X, Y, Z, IX, IY, IZ, PRAD, PANG) will not be selected. To change the a specific component of a location, such as the X coordinate, simply close the calculator by pressing the [Return] key.

Some commands, like the INSERT LINE LINELOC, will continue until the right mouse button or the semicolon key ([;]) is pressed. With INSERT LINE, the end of the first line will be the beginning of the next. By ending the command with a semicolon or the right mouse button, the action is equivalent to "cutting" the continuous line off at the current location.

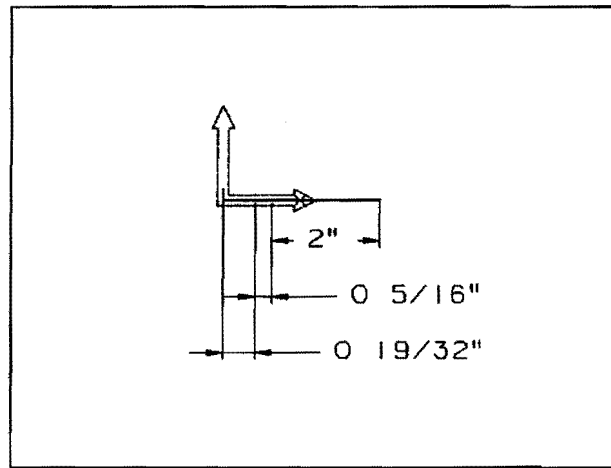


fig. (9) Three Line Segments Created

Step 4: Draw the Arc at the End of the Line Segments

Enter the command sequence:



INSERT ARC ABETWEEN END ent #1 LOC [I] [Y] 7/16*2 [:]

Description: Move the pointer to the INSERT icon in the top left icon pad. The help line will show "INSERT". Press and release the left mouse button to select the icon.

Move the pointer to the ARC icon in the second from the top icon pad. The help line will show "ARC". Press and release the left mouse button to select the icon.

Move the pointer to the ABETWEEN icon in the second from the bottom icon pad. The help line will show "ABETWEEN". Press and release the left mouse button to select the icon.

Move the pointer to the END icon in the bottom icon pad. The help line will show "END". Press and release the left mouse button to select the icon.

Aside: An alternate method of selecting the "END" command is through the [F2] function key.

Move the pointer to the right end of the last line. Press and release the left mouse button. This anchors the beginning of the arc at the end of the line.

Move the pointer to the LOC icon in the bottom icon pad. The help line will show "LOC". Press and release the left mouse button to select the icon.

Aside: An alternate method of selecting the "LOC" command is through the [F1] function key. Press the "I" and the "Y" keys ([I] [Y]). This executes the IY command, and brings up the calculator dialog again.

Note: The LOC command must be active before any of the absolute or relative location commands (X, Y, Z, IX, IY, IZ) can be used.

Enter "7/16*2". This sets the radius of the arc to 7/16 inches, or equivalently, the diameter to 7/8 inches.

Note: The ABETWEEN command uses the diameter of the arc.

Aside: The "/" and the "*" are the divide and multiply commands. These can be entered using the "/" and "*" keys on the keyboard, or using the divide and multiply buttons on the calculator.

Note: The numbers and mathematical operations can be entered using the numeric key pad or the numeric keys at the top of the keyboard.

Press the semi-colon key [;] to end calculator dialog, accept the value and create the arc.

Result:

This creates an arc, or half circle in this case, from the end of the line. The arc has a radius of 7/16 inches.

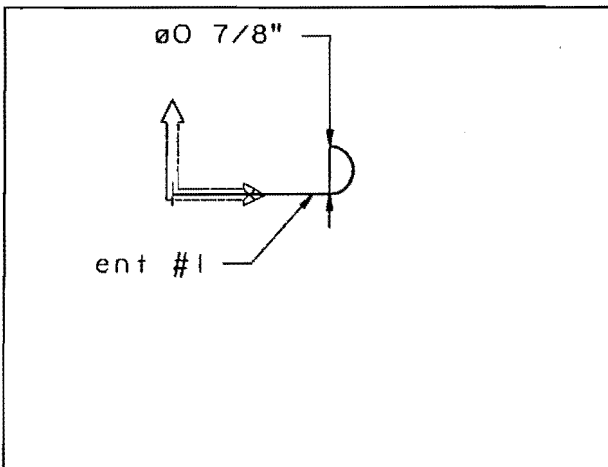


fig. (10) Arc Drawn at End of Line Segments

Step 5: Offset a Line

Enter the command sequence:



TRANSFORM OFFSET DISTANCE 3/4+1 [Return] ent #1 [:] loc #1 [:]

Description: Select the “TRANSFORM”, “OFFSET” and “DISTANCE” commands in sequence using the mouse and pointer. They are in the top, second from the top, and second from the bottom icon pads respectively. When the DISTANCE command is selected, the calculator dialog will appear to accept the offset distance value.

Enter “3/4+1”. This sets the offset distance to 1.75 inches.

Note: The above number cannot be entered as “1+3/4”. That would be interpreted as (1+3)/4, which gives 4/4 or 1.

Note: The value could also be entered as “1.75”.

Press the return key [Return] to end the calculator dialog and accept the value.

Move the pointer to the middle of the middle line segment. Press and release the left mouse button. This selects the middle segment. The segment should appear greyed to indicate that it is selected.

Note: The middle line segment is only 5/16 of an inch long, so care must be taken to select the correct segment.

Note: The delete key [Del] may be used to deselect the last selected entity.

Note: Holding the shift key down while selecting an entity will toggle that entity's selection status.

Press the semi-colon key [;] or the right mouse button to end entity selection, and begin location selection.

Move the pointer to a point about 1 inch above the selected entity and press the left mouse button. A new line segment will appear 1 3/4 inches directly above the selected one.

Aside: Clicking above the entity specified which direction the copy was to be performed in. If the click had been below the selected entity, the new entity would have been put 1 3/4 inches below.

Result:

This draws a temporary construction line which will be used in the next step. This construction line will be deleted later.

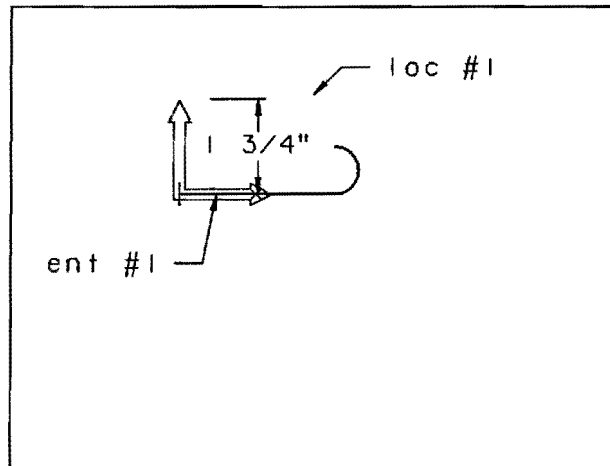


fig. (11) Line Offset

Step 6: Draw the Slanted Line

Enter the command sequence:



INSERT LINE TANPOINT ent #1 END ent#2

Description: Select the “INSERT”, “LINE” and “TANPOINT” commands in sequence using the mouse and pointer. They are in the top, second from the top, and second from the bottom icon pads respectively.

Move the pointer to the top of the arc, press and release the left mouse button. This selects the top of the arc as the tangent entity.

Press the [F2] function key to select the END command.

Move the pointer to the right end of the small line that was created in Step 5, and press and release the left mouse button. This selects the right end of small line entity as the end of the line. A new sloped line segment is created.

Result: This creates a line segment which will be the sloping edge of the bracket. This line is tangent to the arc added in Step 4, and angled up to the construction line created in the previous step.

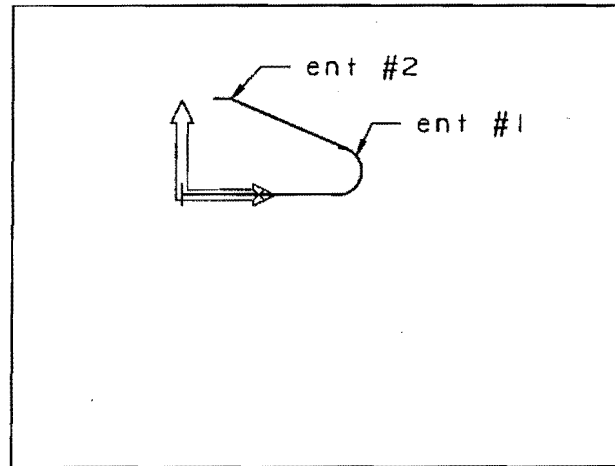


fig. (12) Slanted Line Drawn

Step 7: Erase the Construction Line

Enter the command sequence:



INSERT DELETE ent #1 [:]

Description: Select the “INSERT” and “DELETE” commands in sequence using the mouse and pointer. They are in the top and second from the top icon pads respectively.

Move the pointer to the right end of the small line that was created in Step 5, and press and release the left mouse button. This selects the small line entity.

Press the semi-colon key [:] or the right mouse button to end entity selection and execute the command. This deletes the construction line.

Result: This deletes the small construction line created in Step 5 from the drawing.

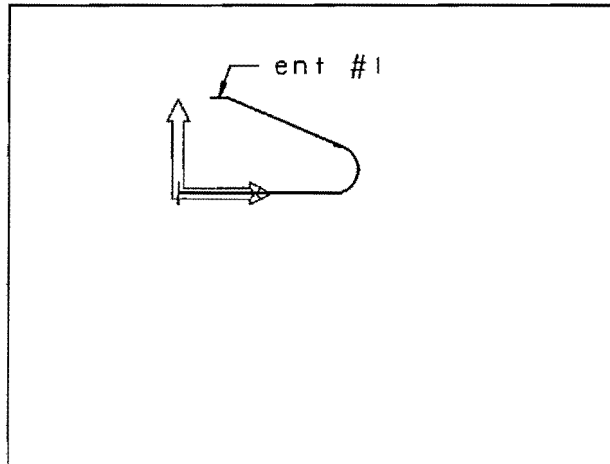


fig. (13) Construction Line Erased

Step 8: Trim the Arc

Enter the command sequence:



TRANSFORM TRIM TRIMINTOF ent #1 ent #2 [;]

Description: Select the “TRANSFORM”, “TRIM” and “TRIMINTOF” commands in sequence using the mouse and pointer. They are in the top, second from the top, and second from the bottom icon pads respectively.

Note: This command shortens or lengthens an entity to its intersection with another entity.

Move the pointer to the upper side of the arc, press and release the left mouse button. This selects the arc as one of the entities to be trimmed.

Note: If the arc is not selected on its upper side, the trim operation will use the lower end of the arc.

Move the pointer to the bottom of the sloped line, press and release the left mouse button. This selects the sloped line as the second entity to be trimmed.

Press the semi-colon key [;] or the right mouse button to end the command and trim the two entities.

Note: If the lower end of the arc is trimmed instead of the upper end, the UNDO command may be used to remove the incorrect change. Simply restart this step after performing the UNDO.

Result:

This trims the upper end of the arc to the intersection of the arc and the sloped line.

Note: The sloped line already had one end point at the intersection of the two entities, so it is unchanged.

Note: The sloped line was created tangent to the arc. Thus there is a single intersection point between the sloped line and the arc.

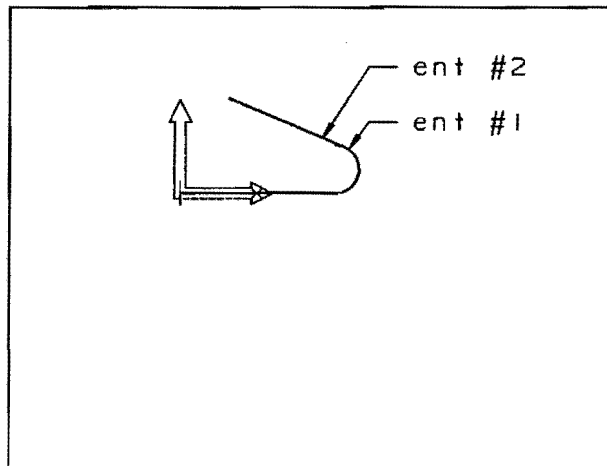


fig. (14) Arc Trimmed

Step 9: Draw the Circle for the Axle

Enter the command sequence:



INSERT CIRCLE CIRDIA 21/64 [Return] ORG ent #1

Description: Select the “INSERT”, “CIRCLE” and “CIRDIA” commands in sequence using the mouse and pointer. They are in the top, second from the top, and second from the bottom icon pads respectively.

When the CIRDIA command is selected, the calculator dialog will appear so that it can accept the diameter of the circle.

Note: This command creates a circle with a specific diameter. It needs a diameter, and a location for the center of the circle.

Enter “21/64”. This sets the diameter of the circle to 21/64 inches.

Press the return key [Return] to end the calculator dialog and accept the value.

Move the pointer to the ORG icon in the bottom icon pad. The help line will show “ORG”. Press and release the left mouse button to select the icon.

Aside: An alternate method of selecting the “ORG” command is through the [F3] function key.

Move the pointer to the arc, press and release the left mouse button. This selects the arc as the entity to used the origin of, and creates the new circle.

Note: This selects the location of the center (or origin) of the arc as the center of the new circle.

Result:

This creates a circle 21/64 inches in diameter, with its center at the same location as the center of the arc. This circle is the hole for the axle of the wheel.

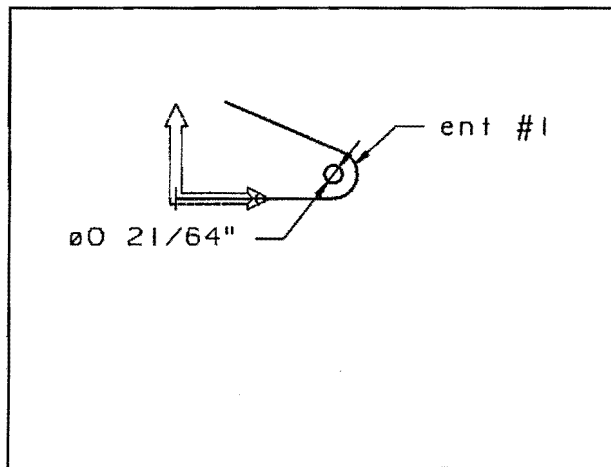


fig. (15) Circle for the Axle Drawn

Step 10: Draw the Circle for the Pivot

Enter the command sequence:



INSERT CIRCLE CIRDIA 29/64 [Return] [Y] 7/8 [.,]

Description: Select the “INSERT”, “CIRCLE” and “CIRDIA” commands in sequence using the mouse and pointer. They are in the top, second from the top, and second from the bottom icon pads respectively.

When the CIRDIA command is selected, the calculator dialog will appear and accept the diameter of the circle.

Enter “29/64”. This sets the diameter of the circle to 29/64 inches.

Press the return key [Return] to end the calculator dialog and accept the value.

Press the [Y] key. This brings up the calculator dialog again. This specifies an absolute location along the Y axis for the center of the circle.

Enter “7/8”. This sets the Y coordinate of the circle center to 7/8 inches. Press the comma key [.,] to end the calculator dialog, accept the value and create the circle.

Note: The value 0 is the default for the X and Z value in this case.

Result: This creates a 29/64 inch circle for the pivot hole.

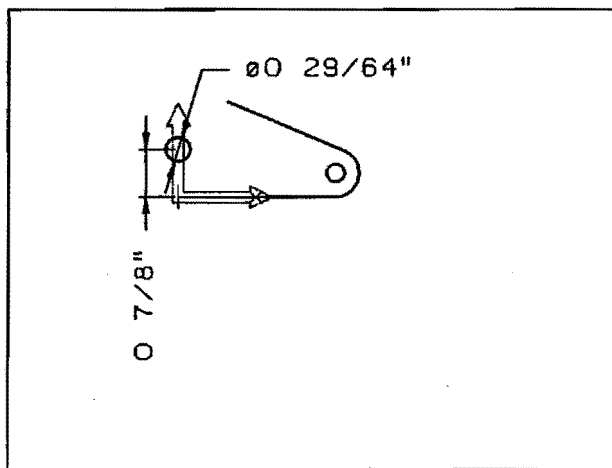


fig. (16) Circle for the Pivot Drawn

Step 11: Change the Plane of the View

Enter the command sequence:



EDITVIEW VIEWGCP Iso view #1

Description: Select the “EDITVIEW” and “VIEWGCP” commands in sequence using the mouse and pointer. They are in the top and second from the top icon pads respectively.

When the VIEWGCP command is selected, the GCP dialog will appear to select the GCP for the view.

Note: This command changes the GCP of a view to a new value.

Move the pointer to the “Iso” button within the dialog. Press and release the left mouse button. The GCP dialog will end, accepting Isometric as the GCP to use.

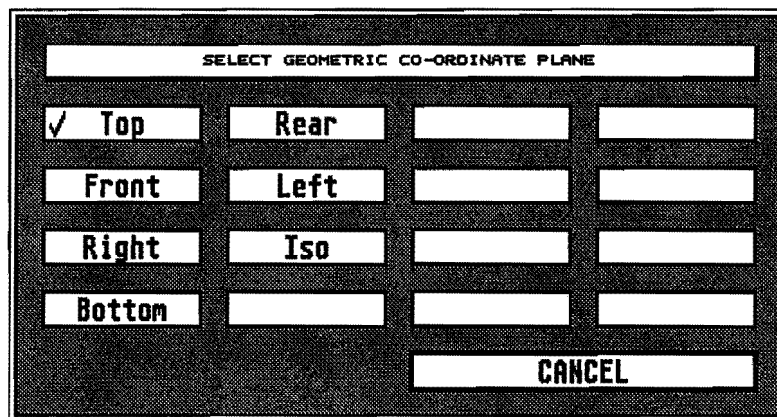


fig. (17) GCP Dialog

Move the pointer to the center of the drawing area. Press and release the left mouse button. This selects the view to be modified. The view immediately rotates to reflect the change in GCP.

Result:

This changes the GCP of the view we are working with from Top to Isometric.

Description of the View:

Up until now, the drawing has looked the same (except for the GCP indicator) as a 2-D drawing. However, we now can see the 3-D effects.

The drawing entities that we have created are now viewed from a different direction. Thus, the view reflects the change in direction by showing the entities from the new perspective.

The GCP indicator now shows all 3 arms. The Z axis arm goes straight up and out of the page. The Y axis arm goes up, to the right and into the page. The X axis arm goes down, to the right, and out of the page.

The GCP indicator shows the current GCP, which is Top at this point. (Remember that the current GCP is also shown in the GCP menu by the check marked menu item.)

Since all the entities were created in the Top GCP, they are on the same plane as the GCP X and Y arms.

Note: The current GCP (Top) and the direction of the view (Iso) are not the same. The current GCP is the plane or viewing direction in which the modifications to the drawing are made.

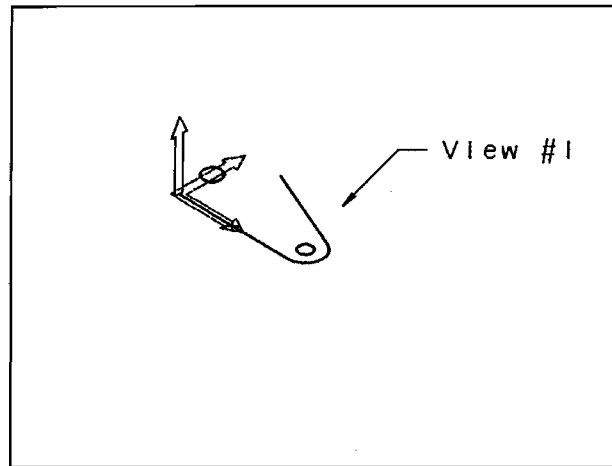


fig. (18) Plane of the View Changed

Step 12: Hide the GCP Indicator

Enter the command sequence:



Menu "Tools", select "GCP" REPAINT

Description: Move the mouse pointer to the Tools menu, press and hold the left mouse button, slide the pointer down to the "GCP" item in the menu so that the item is highlighted, and release the mouse button. This hides the GCP indicator from the drawing display.

Move the pointer to the REPAINT icon in the top right corner of the display. This is the icon second from the right. Press and release the left mouse button. This causes the display to repaint, updating any inconsistencies in the display from the now hidden GCP indicator.

Result:

This hides the GCP indicator, and repaints the display to fix the small inconsistencies caused by hiding the GCP indicator.

Note: Deleting or hiding entities usually creates small display inconsistencies. This is normal, and is simply a display effect.

Note: The REPAINT command can be used at any time and has no effect other than to clean up the display.

Note: An alternate method of selecting the REPAINT command is through the [F10] function key.

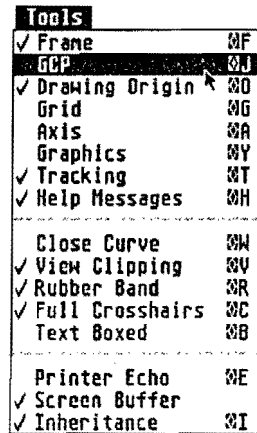


fig. (19) Tools Menu,
Hide GCP

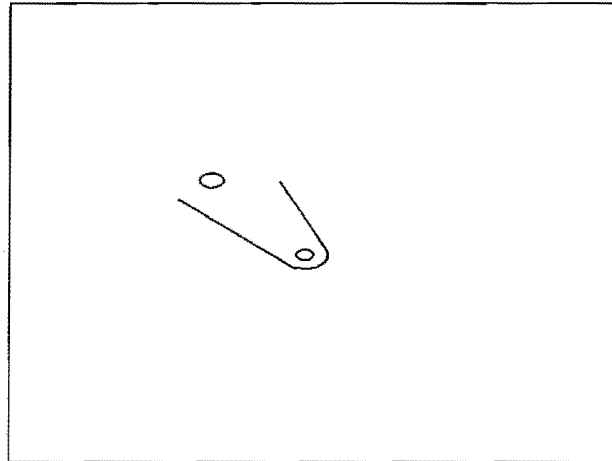


fig. (20) GCP Indicator Hidden

Step 13: Extrude the Entities

Enter the command sequence:



TRANSFORM PROJECT PROJFACE PROJDEPTH 3/32 [Return]
BYALL [;]

Description: Select the “TRANSFORM” and “PROJECT” commands in sequence using the mouse and pointer. They are in the top and second from the top icon pads respectively.

If the “PROJFACE” icon in the second from the bottom icon pad is highlighted, deselect the icon so that it is not highlighted.

Note: Deselecting this icon means that we do not want to create 3D Face entities.

Select the “PROJDEPTH” command from the second from the bottom icon pad. This brings up the calculator dialog, specifying the distance to project the entities along the positive Z axis.

Enter “3/32”. This sets the extrusion distance to 3/32 inches. Press the return key [Return] to end the calculator dialog and accept the value.

Select the “BYALL” command from the bottom icon pad. This selects all the entities in the drawing.

Press the semi-colon key [;] or the right mouse button to end entity selection and extrude the entities.

Result: This makes a copy of all the entities in the drawing, and places those copies 3/32 inches above the originals.

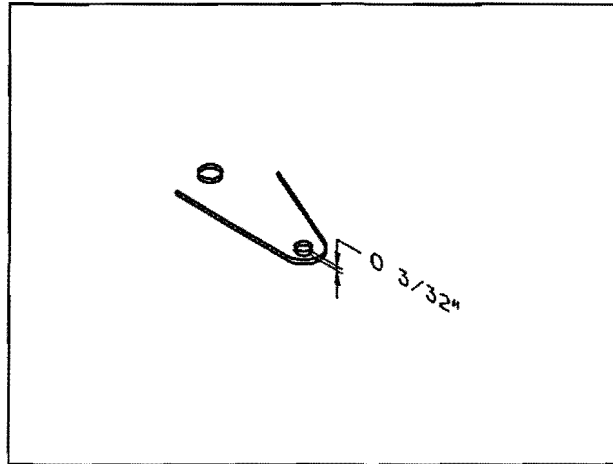


fig. (21) Entities Extruded

Step 14: Bend the Sheet

14.A: Change Current GCP to Front

Enter the command sequence:

Menu "GCP", select "Front"

Description: Select the item labeled "Front" from the menu labeled "GCP".

Result: This changes the current GCP to Front. This is critical to the correct operation of the final part of this step.

The current GCP is now set to Front. All future operations, coordinates and functions will be drawn in this orientation.

Note: The view has its orientation set to Isometric which is different than the current GCP. This is to make selecting the necessary points easier.

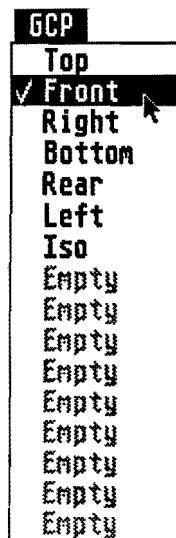


fig. (22) GCP Changed to Front

14.B: Disable TRANCOPY

Enter the command sequence:



TRANCOPY

Description: If the TRANCOPY icon (the first icon on the left in the icon pad along the top of the display) shows two images in the icon, toggle the icon so that it only shows one image. The help message will read "Activate Transformation Copy" when TRANCOPY is disabled.

Result: This disables the copying of entities. The entities are moved in the TRANSFORM commands rather than being copied.

Note: If this command is enabled, the final part of this step will create new entities rather than moving existing ones.

14.C: Zoom Window

Enter the command sequence:



DZWINDOW [Return] 2.75,2.5 [Return] [Return] i3.25,4.25 [Return]

Description: Select the DZWINDOW command from the top right icon pad running along the top of the display.

Note: The [F9] function key is an alternate method of activating DZWINDOW.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "2.75,2.5" and press the [Return] key again. This sets one corner of the zoom window to 2.75,2.5.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "i3.25,4.25" and press the [Return] key again. This sets the second corner of the zoom window to 8.5,6.75. The display will zoom to show a larger view of the drawing.

Note: The "i3.25,4.25" means add 3.25 to 2.75 to get the new location X value and add 4.25 to 2.5 to get the new Y value.

Result: Zoom the display so that the selected window area is completely visible.

Note: The entities in the display are much larger, as the "point of view" that we are using is now closer to the drawing.

Note: The entire page of the drawing is no longer all visible. The zoom has no effect on the actual drawing, but allows us to see features in larger detail.

14.D: Rotate

Enter the command sequence:



TRANSFORM ROTATE ROTANG -90 [Return] BYPWIN [Return]
3.25,4.75 [Return] [Return] 5,5.75 [Return] [Return] 6,4
[Return] [Return] 5.5,3.5 [Return] [;] [;] END ent #1 [;]

Description: Select the commands "TRANSFORM", "ROTATE" and "ROTANG" from the icon pads to the left of the drawing. The "ROTANG" command will bring up the calculator dialog to accept the angle to rotate.

Enter “-90”. This sets the rotation angle to -90 degrees.

Note: Pressing the “=” key will display the value -90 once the “-90” has been entered.

Press the return key ([Return]). This will end the calculator dialog, accepting the values entered in the last step.

Press the [F7] key to select the “BYPWIN” command.

Note: This command selects entities within a polywindow. A polywindow is an irregular polygon defined with three or more points. Only those entities which are totally enclosed within the polywindow will be selected.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “3.25,4.75” and press the [Return] key again. This sets one corner of the poly window to 3.25,4.75.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “5,5.75” and press the [Return] key again. This sets the next corner of the poly window to 5,5.75.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “6,4” and press the [Return] key again. This sets the next corner of the poly window to 6,4.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “5.5,3.5” and press the [Return] key again. This sets the next corner of the poly window to 5.5,3.5.

Press the semi-colon key [;] or the right mouse button to end the polywindow definition and select the entites. Press the semi-colon key [;] again to end the entity selection process and begin the rotation point selection.

Press the [F2] function key to select the END command.

Select the right end of the line which is furthest to the left and on the bottom.

Note: This sets the point about which the rotation will take place.

Press the semi-colon key [;] or the right mouse button to end the location selection and perform the rotation. The selected entities will be rotated downwards.

Result:

The entities selected with a polywindow are rotated in the current GCP, Front. The effect is to bend the sheet along one of its lines.

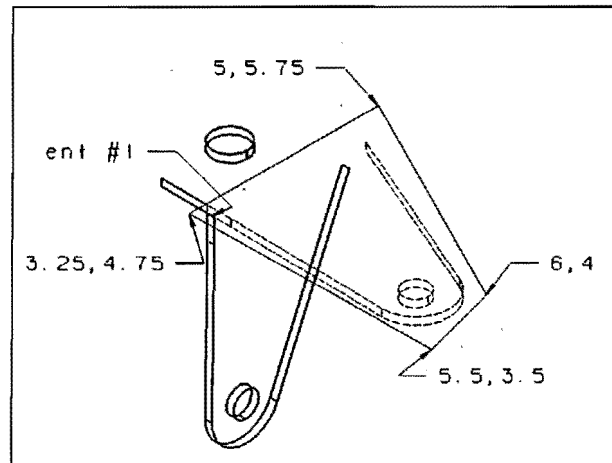


fig. (23) The Sheet Bent

Step 15: Zoom and Erase the Projection Lines

15.A: Zoom Window

Enter the command sequence:



DZWINDOW [Return] 3.25,4.5 [Return] [Return] 5,5.25 [Return] [;]

Description: Select the DZWINDOW command from the top right icon pad running along the top of the display.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "3.25,4.5" and press the [Return] key again. This sets the first corner for the zoom window.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "5,5.25" and press the [Return] key again. This sets the second corner for the zoom window, and zooms the window.

Press the semi-colon key [;] or the right mouse button to execute the DZWINDOW command.

Result: The display will zoom to show a larger view of the drawing. Zooming the drawing will make the entities we wish to work on much larger and more accessible.

15.B: Erase the Projection Lines

Enter the command sequence:



TRANSFORM DELETE ent #1 ent #2 [;]

Description: Select the commands “TRANSFORM” and “DELETE” from the icon pads to the left of the drawing.

Select the entities indicated by the drawing.

Press the semi-colon key [;] or the right mouse button to end the entity selection and perform the entity deletion. The selected entities will be removed from the drawing.

Result: This deletes two of the small projection lines created in Step 13. These two lines would get in the way of the next step.

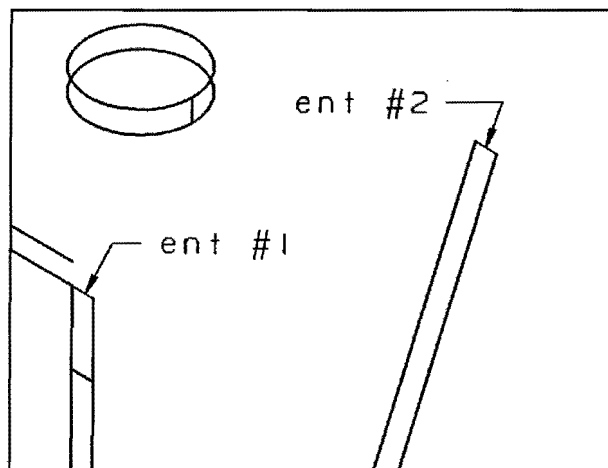


fig. (24) Projection Lines Zoomed and Erased

Step 16: Fillet the Two Edges

Enter the command sequence:



INSERT FILLET FILRAD 1/8 [Return] ent #1 ent#2 [:] ent #3 ent #4 [:]

Description: Select the commands “INSERT”, “FILLET” and “FILRAD” from the icon pads to the left of the drawing.

The “FILRAD” command will bring up the calculator dialog to accept the radius of the fillets.

Enter “1/8”. This sets the fillet radius to 1/8 inches.

Press the return key [Return] to end the calculator dialog and accept the value.

Select the inner two lines, ent #1 and ent #2.

Press the semi-colon key [:] or the right mouse button to end the entity selection and create the fillet. The two selected line entities will be shortened, and the curve of the fillet will join them.

Select the outer two lines, ent #3 and ent #4.

Press the semi-colon key [:] or the right mouse button to end the entity selection and create the fillet. The two selected line entities will be shortened, and the curve of the fillet will join them.

Result: This creates 2 fillets at the upper and lower sides of the corner of the bracket part.

Select the right side of the inner top line and the top side of the inner left line and press the [:] or right mouse button to insert the fillet. Repeat this step for the outer lines.

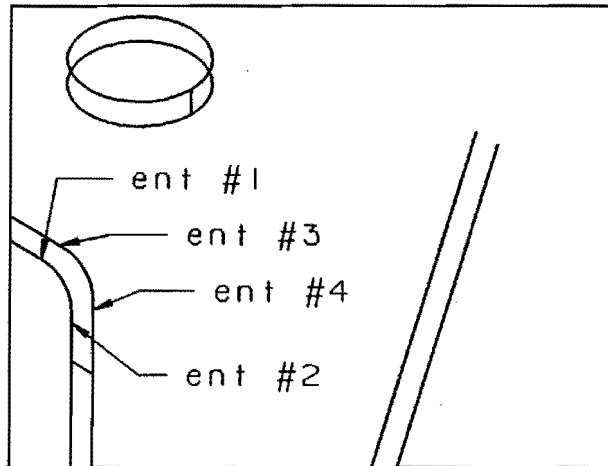


fig. (25) Edges Filleted

Step 17: Draw the Connecting Lines

Enter the command sequence:



INSERT LINE LINELOC END ent #1 ent#2 [;] ent #3 ent #4 [;]

Description: Select the commands “INSERT”, “LINE” and “LINELOC” from the icon pads to the left of the drawing.

Note: The LINELOC is the default command. It will be automatically started.

Press the [F2] function key to select the END command.

Select the two horizontal lines, ent #1 and ent #2.

Press the semi-colon key [;] or the right mouse button to end the continuous line entry.

Select the two vertical lines, ent #3 and ent #4.

Press the semi-colon key [;] or the right mouse button to end the continuous line entry.

Result:

This step draws two lines across the ends of this fillets.

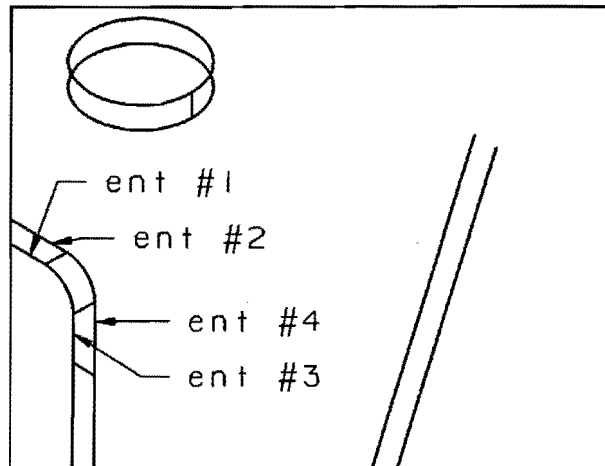


fig. (26) Connecting Lines Drawn

Step 18: Zoom back, Copy Outline to the Other Side

18.A: Zoom Back to the Previous Zoom

Enter the command sequence:



DZLAST

Description: Select the DZLAST command from the top right icon pad running along the top of the display. This zooms the display to the most previous zoom setting before the current one.

Aside: The display will zoom out to the zoom settings of Step 14.A, which were overridden in Step 15.A.

Note: It is possible that the display will not zoom to the desired view. This can occur if this tutorial has not been followed exactly. If this is the case, simply perform the commands specified in Step 14.A.

Result: Zoom the display so that the entities we wish to work on are much larger and more accessible.

18.B: Copy the Outline to the Other Side

Enter the command sequence:



TRANSFORM TRANSLATE TRANCOPY TRANFACE BYWIN
[Return] 2.75,4.25 [Return] [Return] il,1 [Return] [;]
END ent #1 ent #2 [;]

Description: Select the commands “TRANSFORM”, “TRANSLATE” and “TRANCOPY” from the icon pads to the left of the drawing.

Note: There are two TRANCOPY icons, one in the top right icon pad and one in the second from the bottom icon pad. For this command we want the one in the second from the bottom icon pad.

If the TRANFACE icon in the second from the bottom icon pad is highlighted, deselect the icon so that it is not highlighted.

Press the [F6] key to select the “BYWIN” command.

Note: This command selects entities within a window.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “2.75,4.25” and press the [Return] key again. This sets the first corner of the window to 2.75,4.25.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “il,1” and press the [Return] key again. This sets the first corner of the window to 3.75,5.25.

Press the semi-colon key [;] or the right mouse button to end the entity selection.

Press the [F2] function key to select the END command.

Select the outside corners of the indicated lines, ent #1 and ent #2.

Press the semi-colon key [;] or the right mouse button to end the location selection and complete the command. The selected entities are copied to the new location.

Result:

This copies the filleted entities from the front to the back of the bracket. By detailing only the front section of the bracket then copying it to the back, a step has been saved. The other way to achieve this is to detail both the front and the back sections.

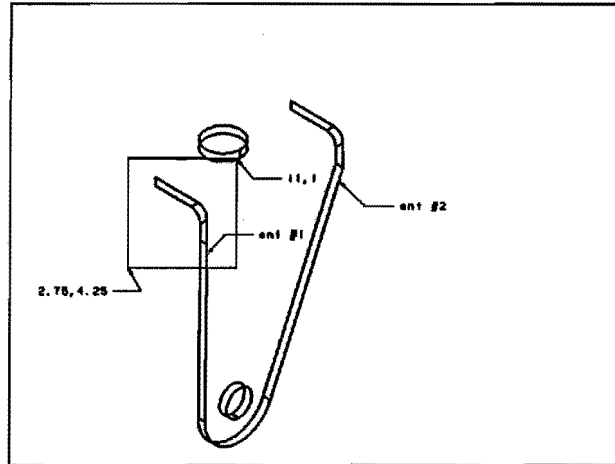


fig. (27) Outline Copied to the Other Side

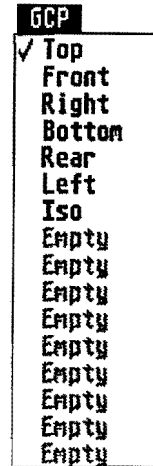
STEP 19: Mirror the Two Sides

19.A: Set current GCP to Top

Enter the command sequence:

Menu "GCP", select "Top"

Description: Select the item labeled "Top" from the menu labeled "GCP".



19.B: Enable TRANCOPY

Enter the command sequence:



TRANCOPY

Description: If the TRANCOPY icon (the first icon on the left in the top right icon pad along the top of the display) shows one image in the icon, select the icon so that the icon shows two images. The help message will read "Deactivate Transformation Copy" when TRANCOPY is enabled.

Result: This enables the copying of entities. The entities are copied in the TRANSFORM commands rather than just being moved.

Note: If this command is disabled, the final part of this step will just move the existing entities rather than creating new ones.

Note: This is the opposite of Step 14.B.

19.C: Mirror the Sides

Enter the command sequence:



**TRANSFORM MIRROR BYWOUT [Return] 3.25,5 [Return] [Return]
i.75,.75 [Return] [:] END ent #1 ent #2 [:]**

Description: Select the commands “TRANSFORM”, “MIRROR” and “BYWOUT” from the icon pads to the left of the drawing.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “3.25,5” and press the [Return] key again. This sets the first corner of the selection window to 3.25,5.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “i.75,.75” and press the [Return] key again. This sets the second corner of the selection window to 4,5.75, and selects the entites outside of the window..

Press the semi-colon key [:] or the right mouse button to end the entity selection.

Press the [F2] function key to select the END command.

Select the ends of the two indicated entities, ent #1 and ent #2.

Press the semi-colon key [:] or the right mouse button to end the location selection and complete the command. The selected entities are copied and mirrored to the other side of the bracket.

Begin selecting the locations at the end of two entities. Select the END command in the locate icon pad or press the [F2] key. Select the top of the indicated entities.

Result:

This mirrors the details from one side to another. This takes advantage of the symmetry between the two sides rather than taking the extra steps to detail each side separately.

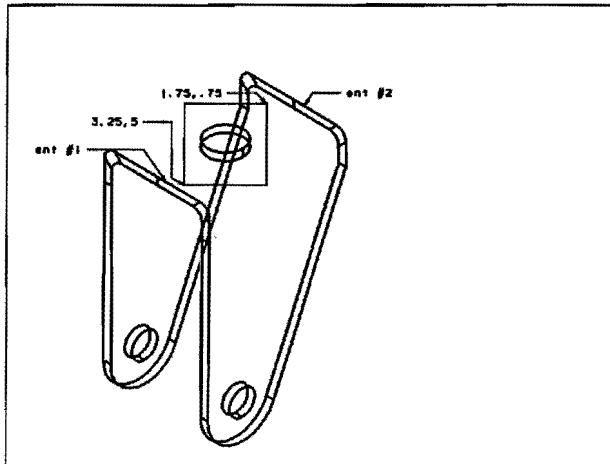


fig. (28) Two Sides Mirrored

Step 20: Zoom and Draw a Construction Line

20.A: Zoom Window

Enter the command sequence:



DZWINDOW [Return] 2.5,3.75 [Return] [Return] 4,2.75 [Return]

Description: Select the DZWINDOW command from the top right icon pad running along the top of the display.

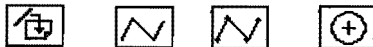
Press the [Return] key to bring up the absolute coordinates dialog. Enter "2.5,3.75" and press the [Return] key again. This sets the first corner of the zoom window to 2.5,3.75.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "4,2.75" and press the [Return] key again. This sets the second corner for the zoom window to 4,2.75. The display will zoom to show a larger view of the drawing.

Result: Zoom the display so that the entities we wish to work on are larger and more accessible.

20.B: Draw the Construction Line

Enter the command sequence:



INSERT LINE LINELOC ORG ent #1 ent #2 [;]

Description: Select the commands "INSERT", "LINE" and "LINELOC" from the icon pads to the left of the drawing.

Note: The LINELOC is the default command. It will be automatically started. Press the [F3] function key to select the ORG command.

Select the two inside circles, ent #1 and ent #2.

Press the semi-colon key [;] or the right mouse button to end the continuous line entry.

To snap to the origin or center of an entity, select the ORG command from the locate icon pad or press the [F3] key.

Result:

The construction line will be used to properly position the entities in their new location.

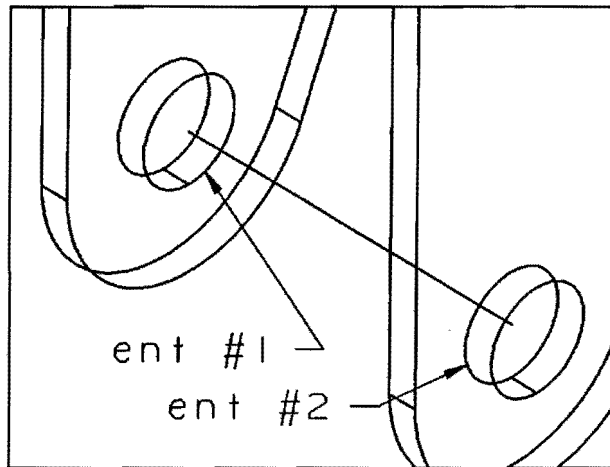


fig. (29) Window Zoomed, Construction Line Drawn

Step 21: Zoom and Move the Entities

21.A: Zoom to Drawing Limits

Enter the command sequence:



DZALL

Description: Select the DZALL command from the top right icon pad running along the top of the display.

Result: This zooms the drawing so that the entire drawing page is visible in the drawing area. This makes the effects of the next part of this step more visible.

21.B: Move the Entities

Enter the command sequence:



TRANSFORM TRANSLATE TRANMOVE BYALL [;] ORG ent #1
LOC [Return] 0,0,0 [Return] [;]

Description: Select the commands “TRANSFORM”, “TRANSLATE” and “TRANMOVE” from the icon pads to the left of the drawing.

Select the “BYALL” command from the bottom icon pad. This selects all the entities in the drawing.

Press the semi-colon key [;] or the right mouse button to end entity selection.

Press the [F3] function key to select the ORG command.

Select the center line, ent #1.

Press the [F1] function key to select the LOC command.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "0,0,0" and press the [Return] key again. This sets the destination offset location to 0,0,0.

Press the semi-colon key [;] or the right mouse button to complete the command and move the entities.

Result:

The bracket part is now positioned for use in the later steps.

The first piece of the caster is now complete.

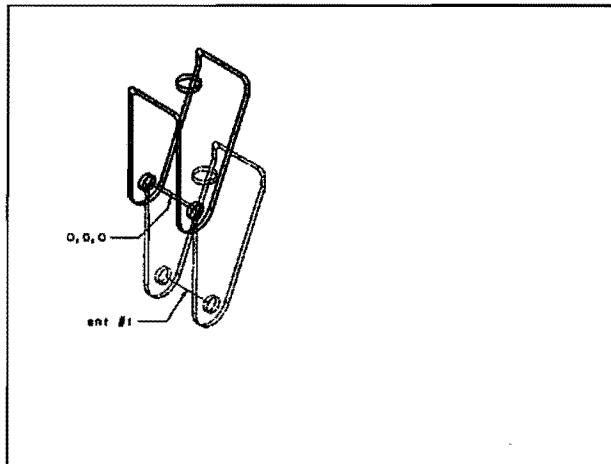


fig. (30) Drawing Zoomed and Entities Moved

Step 22: Select a New Layer

22.A: Select Front GCP

Enter the command sequence:

Menu “GCP”, select “Front”

Description: Select the item labeled “Front”
from the menu labeled “GCP”.



22.B: Select a New Layer

Enter the command sequence:



EDITLAYER SELLYAYER 10 [Return]

Description: Select the commands “EDITLAYER” and
“SELLAYER” from the icon pads to the left of the
drawing.

The “SELLAYER” command will bring up the select
layer dialog to accept the layer name or layer number of
the layer to be selected.

Enter “10” and press the [Return] key again. This
specifies the layer number to be 10.

Note: All the entities in the drawing will become
invisible as they are not on the new layer.

Result:

This selects a new layer (layer number 10) as the current layer. All new entities created in the drawing will be on layer 10. All entities not on layer 10 will be hidden until Step 27, where we turn all the layers on.

Note: The default layer is layer number 0. Thus, all of the previous entities are on layer 0.

Note: We are switching layers to simplify the last few steps of the tutorial.

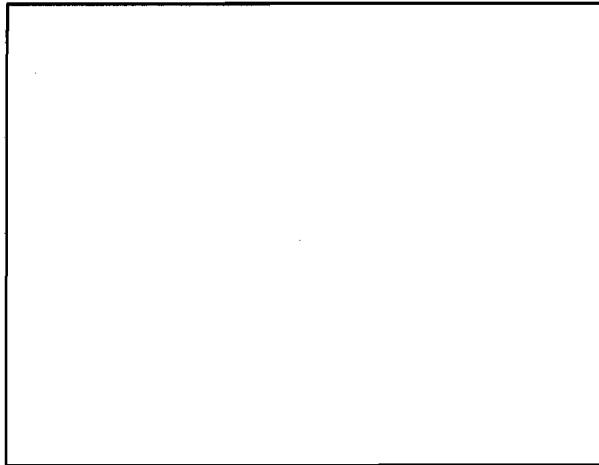


fig. (31) A New Layer Selected

Step 23: Zoom, Draw Horizontal Lines for the Radius

23.A: Zoom Window

Enter the command sequence:



DZWINDOW [Return] 4,6 [Return] [Return] 5.5,3.5 [Return]

Description: Select the DZWINDOW command from the top right icon pad running along the top of the display.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “4,6” and press the [Return] key again. This sets the first corner of the zoom window to 4,6.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “5.5,3.5” and press the [Return] key again. This sets the second corner for the zoom window to 5.5,3.5. The display will zoom to show a larger view of the drawing.

Result: Zoom the display so that the area where we are going to add entities in the next step is more visible.

23.B: Draw the Horizontal Lines for the Radius

Enter the command sequence:



INSERT LINE LINELOC [X] 2 [Return] [Y] 7/16/2 [,] [I] [X] 1/8+1 [,]
[Y] 1/4+1/2 [,] [I] [X] -1/8 [,] [Y] 3/4+2/2 [,] [I] [X] 1/8
[,] [Y] 4/2 [,] [X] 2 [,] [,]

Description: Select the commands “INSERT”, “LINE” and “LINELOC” from the icon pads to the left of the drawing.

Note: The LINELOC is the default command. It will be automatically started.

Press the [X] key, which brings up the calculator dialog. This specifies an absolute location along the X axis for the start of the line segment.

Enter "2" and press the [Return] key. This sets the X value of the location to 2.0.

Press the [Y] key, which brings up the calculator. This specifies an absolute location along the Y axis for the start of the line segment.

Enter "7/16/2" and press the [,] key. This sets the Y value of the location to 7/32, and the comma finishes the line start location entry. The start point of the line is at 2.0, 0.21875.

Note: The Y coordinate value is the diameter of the axle hole, 7/16 inches divided by 2 to get the radius.

Press the "I" and the "X" keys ([I] [X]). This executes the IX command and brings up the calculator.

Enter "1/8+1" and press the [,] key. This sets the length of the line segment along the X direction to 1.125 inches, and creates another line segment. This line represents the width of the caster wheel.

Press the [Y] key, which brings up the calculator.

Enter "1/4+1/2" and press the [,] key. This sets the Y value of the location to 0.625, and creates another line segment. This line represents the lower right side of the wheel.

Press the “I” and the “X” keys ([I] [X]). This executes the IX command, and brings up the calculator dialog.

Enter “-1/8” and press the [,] key. This sets the length of the line segment along the X direction to -0.125 inches, and creates another line segment. This line insets the middle section of the wheel.

Press the [Y] key, which brings up the calculator.

Enter “3/4+2/2” and press the [,] key. This sets the Y value of the location to 1.375, and creates another line segment. This line represents the lower right side of the wheel.

Press the “I” and the “X” keys ([I] [X]). This executes the IX command, and brings up the calculator dialog.

Enter “1/8” and press the [,] key. This sets the length of the line segment along the X direction to 0.125 inches, and creates another line segment. This moves the current location once again to the left side of the wheel.

Press the [Y] key, which brings up the calculator.

Enter “4/2” and press the [,] key. This sets the Y value of the location to 2.0, and creates another line segment. This line represents the upper right side of the wheel.

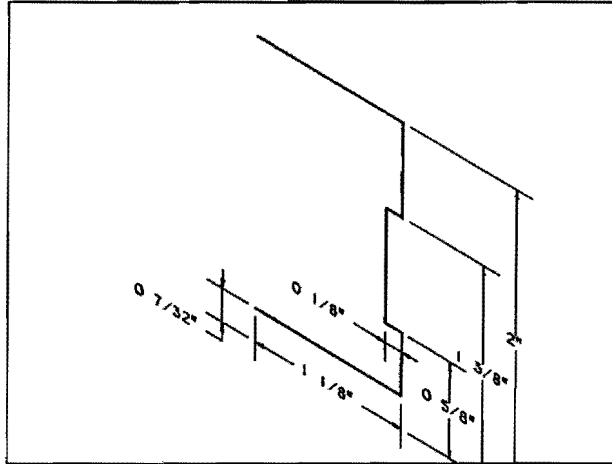
Press the [X] key, which brings up the calculator dialog.

Enter “2” and press the [,] key. This sets the X value of the location to 2.0, and the comma finishes the line start location entry.

Press the semi-colon key [;] or the right mouse button to complete the command.

Result:

Draw the outline of the wheel. The first line is offset from the x axis by the radius of the cut out through the wheel.



*fig. (32) Display Zoomed, Horizontal
Lines for the Radius Drawn*

Step 24: Mirror the Wheel Outline

24.A: Enable TRANCOPY

Enter the command sequence:



TRANCOPY

Description: If the TRANCOPY icon (the first icon on the left in the top right icon pad along the top of the display) shows one image in the icon, select the icon so that it shows two images. The help message will read “Deactivate Transformation Copy” when TRANCOPY is enabled.

Result: This enables the copying of entities. The entities are copied in the TRANSFORM commands rather than just being moved.

Note: TRANCOPY should already be enabled, as it was enabled in Step 19.B.

24.B: Mirror the Wheel Outline

Enter the command sequence:



TRANSFORM MIRROR BYWIN [Return] 5,3.75 [Return] [Return]
5.5,5.75 [Return] [:] ORG ent #1 ent #2 [:]

Description: Select the commands “TRANSFORM” and “MIRROR” from the icon pads to the left of the drawing.

Press the [F6] key to select the “BYWIN” command.

Note: This command selects entities within a window.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “5,3.75” and press the [Return] key again. This sets the first corner of the window to 5,3.75.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "5.5,5.75" and press the [Return] key again. This sets the second corner of the window to 5.5,5.75. This selects all the visible entities within the window.

Press the semi-colon key [;] or the right mouse button to end the entity selection. Press the [F3] function key to select the ORG command.

Select the two indicated entities, ent #1 and ent #2.

Press the semi-colon key [;] or the right mouse button to complete the command and copy the entities.

Result:

Mirror the entities from one side to another. This takes advantage of the symmetry along the center of the wheel of the two sides.

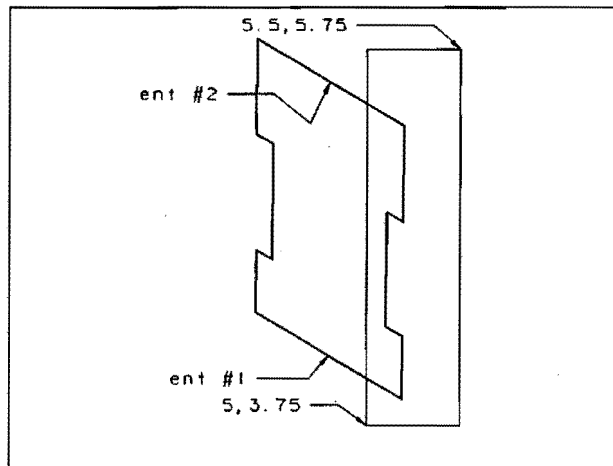


fig. (33) Wheel Outline Mirrored

Step 25: Zoom, Create Outline for the Metal Sleeve

25.A: Zoom Window

Enter the command sequence:



DZWINDOW [Return] 5.5,1.75 [Return] [Return] i3,2 [Return]

Description: Select the DZWINDOW command from the top right icon pad running along the top of the display.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "5.5,1.75" and press the [Return] key again. This sets the first corner of the zoom window to 5.5,1.75.

Press the [Return] key to bring up the absolute coordinates dialog. Enter "i3,2" and press the [Return] key again. This sets the second corner for the zoom window to 8.5,3.75 from the first corner. The display will zoom to show a larger view of the drawing.

Result: Zoom the display so that the area where we are going to add entities in the next step is more visible.

25.B: Create the Outline for the Metal Sleeve

Enter the command sequence:



INSERT BOX [X] 4 [Return] [Y] 21/64/2 [,] [I] [X] 1/8+1 [Return] [Y] 7/16/2 [;]

Description: Select the commands "INSERT" and "BOX" from the icon pads to the left of the drawing.

Press the [X] key, which brings up the calculator dialog. This specifies an absolute location along the X axis for the first corner of the box.

Enter "4" and press the [Return] key. This sets the X value of the location to 4.0.

Note: This is 2 units away from the center of the previous piece.

Press the [Y] key, which brings up the calculator. This specifies an absolute location along the Y axis for the first corner of the box.

Enter "21/64/2" and press the [,] key. This sets the Y value of the location to 21/128, and the comma finishes the entry of the location for the first corner. The first corner of the box is at 4.0, 0.1640625.

Press the "I" and the "X" keys ([I] [X]). This executes the IX command, and brings up the calculator dialog.

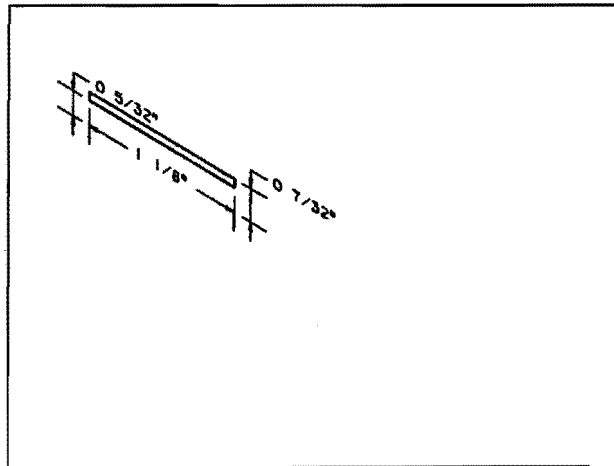
Enter "1/8+1" and press the [Return] key. This sets the X coordinate of the second corner of the box to 5.125 inches.

Press the [Y] key, which brings up the calculator. This specifies an absolute location along the Y axis for the second corner of the box.

Enter "7/16/2" and press the [;] key. This sets the Y value of the location to 7/32. The semi-colon finishes the entry of the location for the second corner, and creates the box. The second corner of the box is at 5.125, 0.21875.

Result:

This draws the outline of the metal sleeve. The outline is a rectangle.



*fig. (34) Display Zoomed, Outline for
the Metal Sleeve Created*

Step 26: Create the Outline for the Joining Bolt

Enter the command sequence:



```
INSERT LINE LINELOC [Return] 6,0,0 [Return] [Y] 5/16/2 [,] [I] [X]
3/32= [M1] 5/8+1- [R1] [,] [Y] 5/8/2 [,] [I] [X] [R1] [,]
[Y] 0 [,] [X] 6 [,] [;]
```

Description: Select the commands “INSERT”, “LINE” and “LINELOC” from the icon pads to the left of the drawing.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “6,0,0” and press the [Return] key again. This sets the line start location to 6,0,0.

Press the [Y] key, which brings up the calculator. Enter “5/16/2” and press the [,] key. This sets the Y value of the location to 5/32, and uses the previous X and Z coordinates.

Press the “I” and the “X” keys ([I] [X]). This executes the IX command, and brings up the calculator dialog.

Enter “3/32=” and press the [M1] button in the calculator. This stores 0.094 in Memory 1 in the calculator.

Enter “5/8+1-”, press the [R1] button in the calculator and press the [,] key. This subtracts the value in Memory1 (0.094) from 1.625 to get 1.531. Press the [Y] key, which brings up the calculator. Enter “5/8/2” and press the [,] key. This sets the Y value of the location to 5/16, and uses the previous X and Z coordinates.

Press the “I” and the “X” keys ([I] [X]). This executes the IX command, and brings up the calculator dialog.

Press the [R1] button in the calculator, and press the [,] key. This uses the value in Memory1 for the X delta.

Press the [Y] key, which brings up the calculator. Enter "0" and press the [,] key.

Press the [X] key, which brings up the calculator. Enter "6" and press the [;] key twice.

Result:

This draws the outline of the metal bolt.

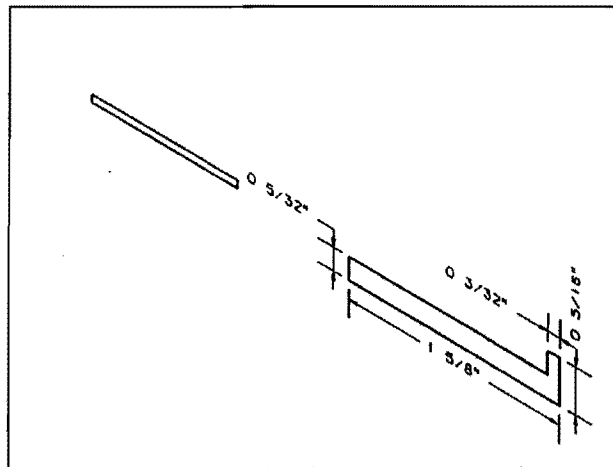


fig. (35) Outline for the Joining Bolt Created

Step 27: Zoom, Revolve the Entities

27.A: Zoom the Drawing

Enter the command sequence:



DZALL

Description: Select the DZALL command from the top right icon pad running along the top of the display.

Result: This zooms the drawing so that only the drawing page is visible in the drawing area.

27.B: Revolve the Entities

Enter the command sequence:



TRANSFORM REVOLVE REVFACE REVANG -270 [Return]
 REVCOPY 27 [Return] BYALL [;] [Return] 0,0,0
 [Return] [Return] 6,0,0 [Return] [;]

Description: Select the commands “TRANSFORM” and “REVOLVE” from the icon pads to the left of the drawing.

Enable the REVFACE icon in the second from the bottom icon pad. It is enabled when it is highlighted.

Select the “REVANG” icon in the second from the bottom icon pad. This brings up the calculator, and it wants the number of degrees to rotate.

Enter “-270” and press the [Return] key. This sets the angle of rotation to -270 degrees.

Select the “REVNCOPY” icon in the second from the bottom icon pad. This brings up the calculator, and it wants the number of degrees to rotate.

Enter “27” and press the [Return] key. This sets the number of copies to create to 27.

Select the “BYALL” command from the bottom icon pad. This selects all the entities in the drawing.

Press the semi-colon key [;] or the right mouse button to end entity selection. Press the [Return] key to bring up the absolute coordinates dialog. Enter “0,0,0” and press the [Return] key again. This sets the line of rotation start location to 0,0,0.

Press the [Return] key to bring up the absolute coordinates dialog. Enter “6,0,0” and press the [Return] key again. This sets the line of rotation end location to 6,0,0.

Press the semi-colon key [;] or the right mouse button to perform the revolution.

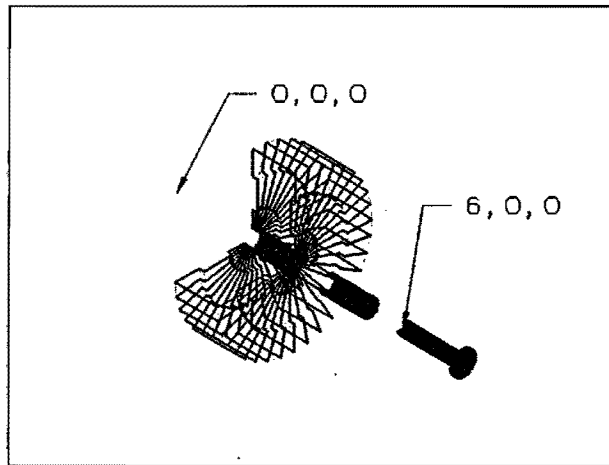


fig. (36) Entities Revolved

27.C: Display the Entities on Layer 0

Enter the command sequence:



EDITLAYER LAYERON 0 [Return]

Description: Select the commands “EDITLAYER” and “LAYERON”. The “LAYERON” command will bring up the select layer dialog to accept the layer name or layer number of the layer to be selected.

Enter “0” and press the [Return] key again. This will make the specified layer “0” to be on.

Result: All the entities are now displayed. This ends the DynaCADD 3-D Tutorial.

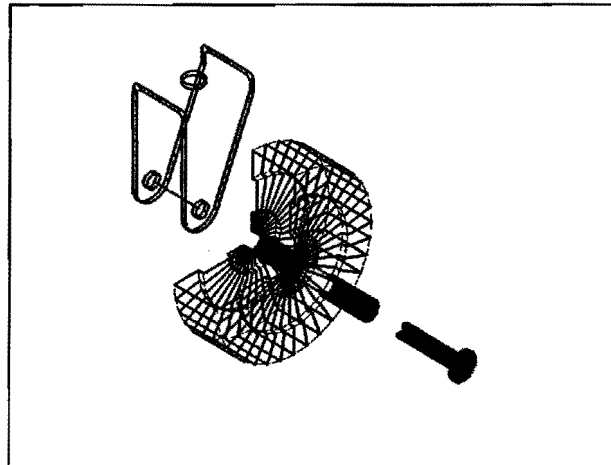


fig. (37) All Entities Displayed



1

2

3

Introduction to DynaCADD ... Chapter 1

DynaCADD is a comprehensive, integrated CADD package. It is very powerful and complex, however, its graphical interface makes it much easier to use than many other CADD programs.

All owners should read the manual thoroughly before attempting to use this package. Even if you have a strong background in CADD, you will benefit greatly from reading through this documentation. For further assistance feel free to contact our Technical Support Department. Telephone (416) 479-2037 or Fax (416) 479-1882. This service is for registered owners only, so please send in your Registration Card today. We ask that you have your manual, questions and program serial number handy when you call.

This manual assumes that you have a working knowledge of your computer system and mouse. If you have any questions relating to these, please refer to the respective manual supplied with your computer or mouse. It is also important that you understand the use of folders (sub-directories), paths and drives.

Backing up DynaCADD

For your convenience, we have not copy-protected the DynaCADD disks. We allow our registered owners to backup DynaCADD under the terms and conditions specified in the License Agreement, located at the back of the manual. Please backup the DynaCADD disks now and place the originals in a safe place. Refer to your operating system manual for exact instructions on how to format and copy disks.

README File

Before you continue, please check DynaCADD Disk 1 for a file called README. This text file contains any new information and update material relevant to the program.

Installing DynaCADD

In order to run DynaCADD, your 80286/80386/80486 based MS-DOS/PC-DOS system requires a standard graphics card with a minimum resolution of 640 by 350, 640k of RAM, a minimum of 1 megabyte of

Introduction to DynaCADD®

expanded memory (EMS 3.2 or 4.0), a hard drive, and a two, or more, button mouse. A math coprocessor is also strongly recommended.

On 80286 based systems, expanded memory needs to be supported via hardware. Strictly software based expanded memory emulators will not work properly with DynaCADD. Consult your computer supplier or manufacturer for details on your 80286 based machine and its expanded memory support.

On 80386 and 80486 based systems, expanded memory managers such as QEMM from Quarterdeck Office Systems work well with DynaCADD. Please consult the respective memory manager manual for installation instructions.

DynaCADD is available in two formats: two 5 1/4" 1.2 Mb or two 3 1/2" 1.44 Mb disks:

- Disk 1 contains the INSTALL utility, GEM/3 run-time and DynaCADD.
- Disk 2 contains the Font Editor, Make Plot, fonts, drivers and tutorial drawing files.

To Install DynaCADD

In the installation program use the up and down arrow keys to move the selection bar. The [Esc] key can be used to back up to the previous dialog.

1. Insert Disk 1 into drive A.
2. Type A: and press [Enter].
3. Type INSTALL and press [Enter].
4. Press [Enter] with the selection bar over the "Install GEM/3 into System" option.
5. Highlight the source drive letter and press [Enter].
6. Highlight the drive on which you wish GEM/3 to be installed and press [Enter].
7. Highlight the appropriate graphics card with the selection bar and press [Enter].

8. Highlight the mouse or tablet type with the selection bar and press [Enter].
9. A menu may pop up asking which serial port the mouse or tablet is hooked up to. Highlight the appropriate port and press [Enter].
The installation program will now proceed to copy the GEM/3 run-time and the suitable screen driver to your hard disk. When it is completed a message will appear.
10. Move the selection bar to "Install DynaCADD into System" and press [Enter].
11. Type in your name and address pressing [Enter] at the end of every line. Press [Enter] through all five lines.
12. Highlight the source drive letter and press [Enter].
13. Highlight the drive on which you wish DynaCADD to be installed and press [Enter].
14. INSTALL will now copy the contents of Disk 1 to your hard disk.
15. When INSTALL is finished with Disk 1, it will ask for Disk 2. Insert Disk 2 into drive A and press [Enter]. INSTALL will now copy the contents of Disk 2 to your hard disk.

DynaCADD is now installed on your system.

About This Manual

This manual is divided into three basic parts:

- General Information and Basic Theory (chapters 1 to 5)
- Command and Technical Reference (chapters 6 to 16)
- Appendices and Glossary

We recommend that you read chapters 1 through 5 before attempting to use DynaCADD. If you have no previous experience with CADD, we also recommend that you read and follow the 2-D tutorial.

The reference section follows the basic order of the pull down menus and icon pads as they appear on the screen. Each command is individually described with information given on all relevant modes, flags and modifiers. Actual commands and modifiers are shown in uppercase.

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In the command descriptions we use abbreviations to refer to entities and selections. As an example: loc #1, loc #2, etc., refer to locations and ent #1, ent #2, etc., refer to entities.

Flags are a form of optional modifier such as an angle or distance. The flag value is entered through the pop up calculator. Flags also have default settings that are used if you do not change them. If you do enter a value, it becomes the new default and is used with that command from then on. Look for flags at the Primary Command description.

If you forget what entities are valid for a particular command, click on the entity FILTER icon on the bottom icon pad. All valid entities for the current command are displayed in the dialog. Any invalid entries are disabled.

Keyboard Conventions



Throughout the manual you will see references to [Return], [Help] and [Undo]. [Return] and [Enter] are interchangeable. [Help] refers to [F11] or [Page Down]. [Undo] refers to [F12] or [Page Up].

In this manual, keys are noted in square brackets. For example:

[Enter] [Tab] [,] [:] [Space] [Esc] [Ctrl-A]

When you see any of the above it means to press the particular key, or, the combination of keys together.

Semicolon [:]: Used frequently in DynaCADD commands to indicate where a selection is complete and command execution should continue. Pressing [:] is equivalent to pressing the right mouse button.

Comma [,]: Shown on the lower icon pad, the comma command is used to separate data in a command. Do not confuse it with the semicolon.

Alternate [Alt]: Most menu commands can be activated by pressing the [Alt] key together with another key. The key is shown beside the menu option with a diamond symbol. For example: [Alt-S] will save the current part and drawing.

Control [Ctrl]: You can create your own keyboard macro commands and assign them to Control key combinations. See Chapter 6 for details.

Function keys: The function keys have several commonly used commands pre-assigned to them. These assignments cannot be changed. See Chapter 4 for details.

Case is unimportant when you enter commands or layer names. Everything typed as a command is automatically converted to uppercase.

Mouse Conventions

There are several basic mouse functions referred to in this manual:

Click: Press the mouse button once. Unless otherwise noted, this command refers to the left mouse button. Clicking the right mouse button is equivalent to pressing [;], which is mostly used to indicate that the selection process is complete.

Click is often used to enter a command and to select an entity or location.

Double click: Click the left mouse button twice in quick succession.

Drag: Press and hold the left mouse button while you move the mouse. This “grabs” an object which is moved with the pointer. Release the mouse button to let go of the object at the new location.

Select: Choosing an option or clicking on a button to turn the feature on. Many dialog buttons are toggles; that is if you click them once (select them) they are turned on. A second click turns them off.

In the File Selector you can click on a file name and then click on OK to select it, or, you can double click on the file name to achieve the same result.

The mouse cursor is usually referred to as the “pointer”. Normally it is in the form of an arrow. However it may also be a pointing hand, an open hand, or another symbol. In the drawing area the cursor is a crosshair.

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Do not confuse instructions intended for “buttons” displayed in dialogs with those intended for the mouse buttons. The intent should be clear in the context of the documentation.

Editing Dialog Text

In DynaCADD there are several dialogs where you enter or edit text. Use the following keys for editing:

- | | |
|--------------|--|
| [Esc] | Clears the entry. |
| [Backspace] | Erases the character to the left of the cursor. |
| [Delete] | Erases the character to the right of the cursor. |
| [Arrow keys] | Move the cursor in the indicated direction. |
| [Enter] | Accepts the entry and closes the dialog. |

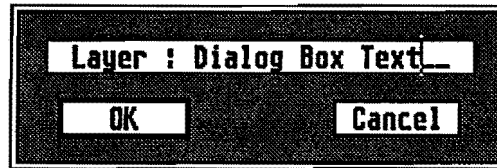


fig. (1) Editing Dialog Text

The Text Editor

The text editor is a basic utility in which you can enter and edit text for several commands. The maximum size of text allowed is 800 characters or one window. The keys used in the text editor for editing are:

- | | |
|-----------------------------|--|
| [Backspace] | Erases the character to the left of the cursor. |
| [Delete] | Erases the character to the right of the cursor. |
| [Up arrow] | Moves the cursor to the end of the previous line. |
| [Down arrow] | Moves the cursor to the start of the next line. |
| [Left arrow] | Moves the cursor one character to the left. |
| [Right arrow] | Moves the cursor one character to the right. |
| [Home] | The first press moves the cursor to the start of the line.
The second press, or if the cursor was already at the start of the line, moves the cursor to the start of the text. |
| [End] | The first press moves the cursor to the end of the line.
The second press, or if the cursor was already at the end of the line, moves the cursor to the end of the text. |
| [Enter] | Inserts a carriage return, ends the line and moves the cursor to the start of the line below. |
| [Undo] | Cancels the editing and returns to the drawing. |
| [Insert] | Accepts the entry and closes the text editor. |
| [F11] or [Page Down] | Displays help and information about the text editor. |
| %%nnn | This allows you to enter special characters which cannot be typed directly or easily through the keyboard. Type %% followed by the three digit number of the special character. The character number must be padded with zeros and have a value between 1 and 255. For example: the hundred and thirty-fourth character of a font is inserted by typing %%134. |

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The following special characters used in dimensioning can be entered using a short hand form:

diameter sign %%c

degree sign %%d

plus/minus sign %%p

For more information and a list of special characters see Appendix H.

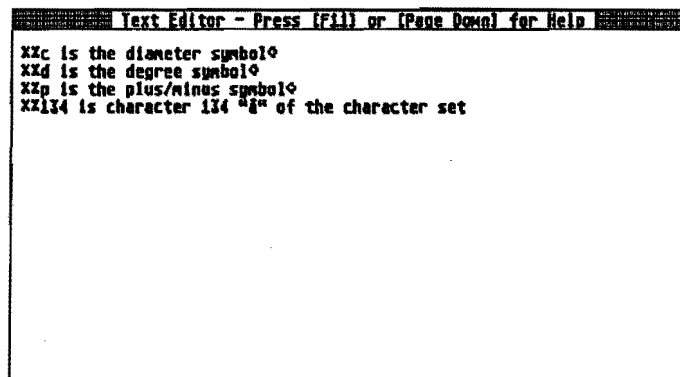


fig. (2) Text Editor

Translations

DynaCADD has the ability to translate other CADD formats into DynaCADD:

DXF: DXF files, used in AutoCAD and several other CAD programs, can be read in and written out using the File Transfer menu selection. DXF file conversions are discussed in Appendix A of this manual.

DEF: DEF is DynaCADD's own ASCII file format for use with other programs. DEF files can be read and written out by using the File Transfer menu selection. The DEF file format is described in detail in Appendix A of this manual.

Fonts

DynaCADD comes with a sophisticated vector font editor described in Chapter 16. With this editor you can create and modify your own fonts. Several high quality Compugraphic® fonts are also provided with DynaCADD. These can be loaded into DynaCADD but cannot be edited in the vector font editor.

Loading DynaCADD

Before loading, DynaCADD must be fully installed. Please see Installing DynaCADD if DynaCADD has not yet been installed on your system.

To load DynaCADD:

1. From the command prompt type C: and press [Enter].
2. Type DYNACADD and press [Enter]. This will bring you to the DynaCADD system level.
3. Select Activate Part. Choose a part (.PRT) in the parts folder.
4. Select Activate Drawing. Choose a drawing (.DWG).
5. Click on OK. This will now bring you to the CADD level.

The first time that you run DynaCADD, you enter the program at the System level (see the next chapter). After that, DynaCADD keeps track of where you left off the last time a drawing was saved. When DynaCADD is run again, a dialog appears. You can choose to return to the previous session by clicking on AUTO, start at the System level by clicking on NO, or quit the program by clicking on EXIT.

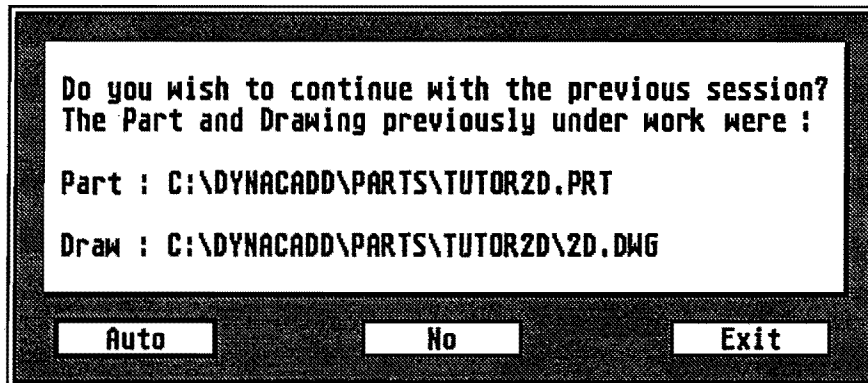


fig. (3) DynaCADD® Auto Start Dialog

System Level Introduction ... Chapter 2

DynaCADD is divided into two distinct levels:

System level: The level at which you first enter into DynaCADD. Here you perform the initial set up of parts and drawings as well as specification of parameters such as scale, units and page size. Whenever you want to change these parameters in a design session, you return here.

System level consists of a large dialog with the options placed in the approximate order in which you would normally use them.

CADD level: You enter the CADD level after the first time you load DynaCADD and set the System level parameters. The CADD level is used for all design work, geometry creation and modification. This is where you create your drawings and send them to various output devices.

Before continuing with the explanation of the System level dialog, let us look at the terminology used in this chapter.

Part: A three dimensional database that defines the basic elements (an entity or a group of entities; see below) in your drawing. Think of a part as an actual object held within the memory of the computer. A part can be manipulated, copied, moved or modified in almost any way you desire.

Drawing: The actual page or sheet of paper on which you are working. You specify the size and scale for the drawing page in the System level. A part can have more than one drawing associated with it.

System Level Introduction

MODIFY CURRENT DRAWING PARAMETERS									
DRAWING NAME / DRAWING UNITS					DRAWING SIZE				
Activate Part...					A B C D E				
Activate Drawing...					AA AB AC AD AE				
					A4 A3 A2 A1 A0				
					W : 17.00000 H : 11.00000				
DRAWING SCALE									
D:0001					[In] Ft Vd Hs mm cm m km				
A:0001					[In] Ft Vd Hs mm cm m km				
OK									

fig. (4) System Level Dialog

Setting Current Drawing Parameters

The normal procedure for setting up the part and drawing is as follows:

1. Activate a part.
2. Activate a drawing.
3. Set the drawing size (optional).
4. Set the drawing units (optional).
5. Set the drawing scale (optional).
6. Enter the CADD level.

You seldom need to return to this dialog, only to change the default set up parameters or switch to a different part or drawing. Every time you load DynaCADD, you are asked if you want to return to the point at which the previous session was last saved with all the parameters intact. If you select no, you enter the System level.

Whenever you want to load a new part or drawing, or change scale or units, you must select drawing from the Set menu to go back to the System level and perform the appropriate actions.

Activating a Part

This loads a part from disk into the drawing. Use the mouse to move the pointer over the Activate Part button on the screen. Click the left mouse button. A standard File Selector dialog appears on the screen. Type in the name of the part you wish to activate or select a part file from the list. You can place a part file in any directory on any drive. The .PRT extension is automatically added to the part filename should you forget.

Activating a Drawing

This loads a drawing from disk. Move the pointer over the Activate Drawing button and click. You must place the drawing filename in the same directory at which the File Selector dialog opened. DynaCADD does not accept a drawing in any other directory. The .DWG extension is automatically added to the drawing filename should you forget.

Setting Drawing Units

This determines the units of measurement used in the drawing.

The computer needs to relate the values and size of your drawing to the real world. For example, if you tell the computer to draw a circle with a diameter of 4.0, the computer must know how large to draw the circle when it comes time to plot or print the drawing. There are eight predefined units available for use in the DynaCADD package:

English:	Inches (In)	Metric:	Millimeters (mm)
	Feet (Ft)		Centimeters (cm)
	Yards (Yd)		Meters (m)
	Miles (Mi)		Kilometers (km)

Setting the Drawing Scale

This sets the relationship between the screen units and the real world the drawing represents.

There are times when it is not practical to plot out a drawing at its actual size. If, for example, a civil engineer has designed a bridge and wants to plot the drawing, it is neither practical nor possible to plot the whole drawing out at a constant 1:1 ratio.

In order to fit large designs onto a workable sheet of paper, a drawing scale is used. DynaCADD uses a scaling relationship as a means of sizing a drawing up or down. There are two buttons, each with eight unit buttons, used to set the drawing scale: one labelled "D" for drawing size, the other labelled "A" for actual size. A few examples should clarify the use of these buttons:

Setting a Drawing at 1/2 scale:

1. Click on the "D" button.
2. Press [Esc] to clear the existing value and type "1".
3. Click on the "A" button.
4. Press [Esc] to clear the existing value and type "2".
5. Ensure the unit type buttons beside the "A" and "D" button are equivalent.

Setting 1/8 inches equal to 1 foot:

1. Click on the "D" button.
2. Press [Esc] to clear the existing value and type "1".
3. Click the "In" (Inches) button to the right of the "D" button.
4. Click on the "A" button.
5. Press [Esc] to clear the existing value and type "8".
6. Click the "Ft" (Feet) button to the right of the "D" button.

The drawing scale is interpreted to read:

1 inch on the drawing represents 8 actual feet.

Setting the Drawing Size

This determines the physical limits of the drawing for both design and printing.

There are two methods for setting the size of the page:

Method 1

This is the simplest of the two methods. Simply click on any one of the fifteen predefined size buttons, labeled A to E, A4 to A0 and AA to AE to reflect the English, Metric or Architectural standard sizes, respectively. The new drawing size appears beside the width and height labels in the main dialog. The predefined sizes, widths and heights are:

English:

A.	11.0 in. by 8.5 in.
B.	17.0 in. by 11.0 in.
C.	22.0 in. by 17.0 in.
D.	34.0 in. by 22.0 in.
E.	44.0 in. by 34.0 in.

Architectural:

AA.	12.0 in. by 9.0 in.
AB.	18.0 in. by 12.0 in.
AC.	24.0 in. by 18.0 in.
AD.	36.0 in. by 24.0 in.
AE.	48.0 in. by 36.0 in.

Metric:

A4.	21.0 cm. by 29.7 cm.
A3.	29.7 cm. by 42.0 cm.
A2.	42.0 cm. by 59.4 cm.
A1.	59.4 cm. by 84.1 cm.
A0.	84.1 cm. by 118.9 cm.

System Level Introduction

Method 2

If the drawing size you want is not among the above selections, you may specify any width and height combination you prefer. To specify a new width, click on the Width button. A small text cursor appears at the end of the current width value. Use [Backspace] or [Esc] to clear the entry and type a new width. Keep in mind that the width you specify is always in the current units selected (see below). Use the identical procedure to specify the height of the drawing.

Entering CADD Level

This is where you enter the design level of DynaCADD.

The OK button is enabled provided the following conditions have been met:

- a part is active
- a drawing is active
- a valid scale has been specified
- a valid width/height ratio has been specified

To enter CADD level, click on the OK button or press [Return].

If you selected a part and/or drawing on the disk, the files are now loaded. The system level dialog disappears and the selected part and drawing is displayed. You may return to the system level dialog at any time by pulling down the Set menu and selecting the Drawing option.

Basic Theory ... Chapter 3

It is very important that all owners of DynaCADD, regardless of previous experience, read and understand this chapter. Some of the terms described here are unique to DynaCADD, although the concepts they represent are universal to professional level CADD.

Connected to any specialized field are numerous terms or buzzwords familiar only to those who work in that particular field. CADD is no exception. Since DynaCADD allows both two and three dimensional drafting and design, there are even more of these terms to learn. The purpose of this chapter is to give you an understanding of the terms used throughout this manual.

Entity

An entity is the basic geometric object in DynaCADD. It is created or modified when you use the INSERT or TRANSFORM commands. INSERT is used to create new entities, TRANSFORM lets you modify existing entities. Once inserted or placed, you can modify and manipulate an entity in numerous ways, including grouping it with other entities to create larger, complex parts. In their basic form, an entity is any of these primitives:

- Point
- Line
- Circle
- Arc
- Fillet (see below)
- Ellipse
- Elliptical arc
- Text
- B-spline curve
- Bézier curve
- 2-D filled solid
- 3-D face

By combining these entities, you can create and modify complex parts and drawings quickly and easily. Entities can be broken into pieces, each one an individual entity. Most entities cannot be joined together to create single entities, except as a group (curves can be joined; see Chapter 8). In some commands, several entities may be selected at once and acted upon in concert.

Not all entities are valid for every command. For example, you cannot stretch a point. The entity **FILTER** dialog lists all entities valid for every command (see Chapter 5).

Remember that although you can build an object out of several entities, unless formed into a group, the entities remain as individual components that can be moved, deleted or modified separately, without affecting the position or attributes of any other entities in the drawing.

A **FILLET** has its own **INSERT** commands and is referred to throughout the manual as a separate entity type for clarity. However, it is in all aspects an **ARC** and where an **ARC** is a valid entity, so is a **FILLET**.



2-D Entities vs. 3-D Entities

DynaCADD has two main design modes, 2-D and 3-D. You can switch between them at any time by clicking the mode (**INIT2D/INIT3D**) toggle icon. 2-D and 3-D entities can exist on a drawing simultaneously but some commands are only relevant to one mode, not both. For example, **VIEW** and **GCP** commands affect 3-D entities only.

There are times when it is not convenient to work in three dimensional space. To simplify the design process, you can specify if you wish the entities you are creating as either three dimensional or two dimensional. Both can be seen on the drawing at the same time but you can only work on or create 2-D entities in 2-D mode and 3-D entities in 3-D mode.

Two dimensional entities are “flat” and lie on the plane of the screen; the process is similar to drawing on paper with a pencil. They are defined by a simple X-Y coordinate system that extends from the page origin (see coordinate systems, below). When you switch to 3-D mode, 2-D entities remain in their same orientation, parallel to the drawing screen, no matter what viewing plane the 3-D view is using.

Three dimensional entities are placed in space and defined by an X-Y-Z coordinate system (see coordinate systems, below). You can rotate these objects in space, from within the design process, or rotate your view about them to see them from different locations. Unlike 2-D entities, 3-D entities are much more than pencil marks on paper. Think of them as actual objects that exist in depth behind the drawing on which you are working.

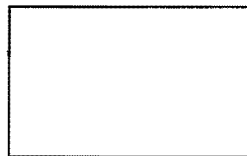
In order to see and manipulate 3-D entities, you must open a "View" on the drawing (see below). With different views on the same drawing, you can see the full scope of a 3-D part.

3-D entities do not automatically have three dimensions when they are created. They are inserted into the current view like 2-D entities. In order to add depth, you must "project" them into the third dimension by using the TRANSFORM commands (see Chapter 10).

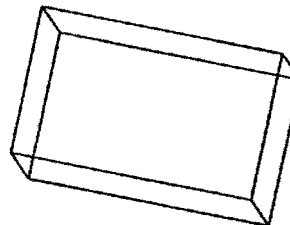
Views

Views are a means to see your drawing from different perspectives when designing a three dimensional part. As noted previously, 3-D entities have depth behind the drawing on which you are working. You can open a "window" to the drawing and use that window, a view in DynaCADD terminology, to look into the three dimensional space that exists behind your drawing.

Views are, however, much more than just regular windows. You can specify how far they should step back when looking at the part and what rotation and scale they use. You can scroll views in order to see various sections of the part. In addition to all this, DynaCADD allows you to have up to four of these views on any one drawing, each looking at a different part of your database.



2 Dimensional Box



3 Dimensional Cube

fig. (5) 2-D Entities vs. 3-D Entities

Basic Theory

In 3-D mode, you must have at least one view open.

During the design process, you can work in any combination of views at any time. This means that you can see your part rotated at four different angles and zoomed on four different areas simultaneously.

Keep in mind that although you can see four different views of one part, it is still only a single part. Any changes you make in one view are instantly reflected in all views.

Another common use of views is the automatic generation of orthographic views. With traditional drafting, a part is represented by showing numerous two dimensional orthographic representations of the object (ie. Top view, Front view and Right side view). By combining the information contained in each individual view, you can mentally piece together what the actual object looks like. DynaCADD has the ability to create these 2-D representations automatically, anywhere on the drawing, at any scale. One of the major advantages of 3-D design work is the ability to modify a part once and have all orthographic views instantly updated.

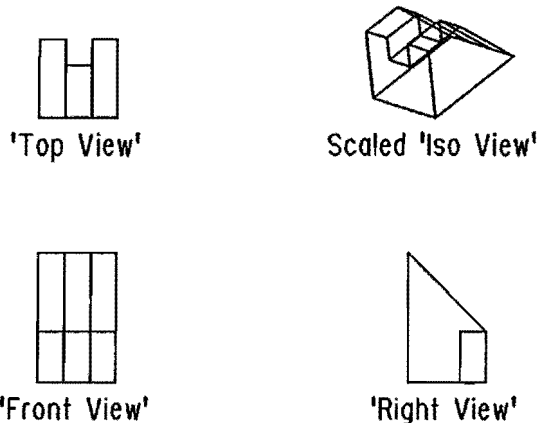


fig. (6) Four Views Active

The most important points to remember about views are:

- Views are a means of looking at the 3-D part you are creating.
- You can open up to four views on the drawing at any one time.
- You may work in all views simultaneously.
- Multiple views show a single part from different angles.
- Each view has its own zoom factor, scale, 3-D rotation, scroll factor and drawing orientation (position on the drawing).
- Views are a simple means of generating multiple orthographic views.

Coordinate Systems

Since DynaCADD allows the creation of 2-D and 3-D entities, each type must have its own coordinate system. The 2-D coordinate system works from the “page origin”, initially located in the lower left corner of the drawing. The 3-D system is much more complex and uses what we refer to as Geometric Coordinate Planes (GCPs).

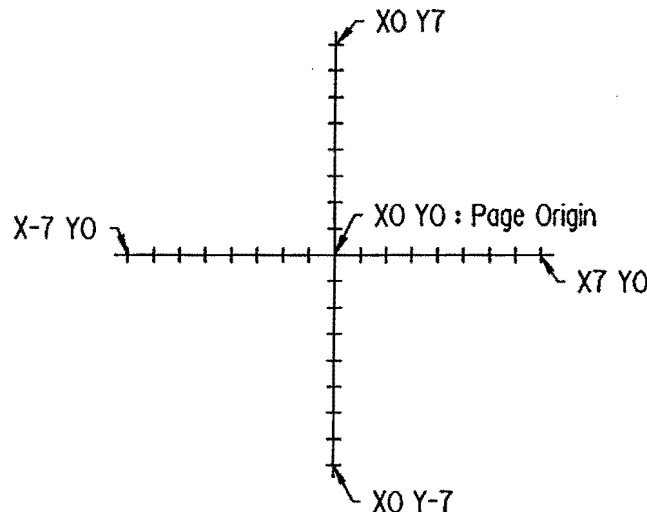


fig. (7) 2-D Coordinate System

Page Origin

The page origin is the 0, 0 position on a 2-D drawing, graphically displayed as a small box with a point in the center. The drawing axis runs through this location with the positive X axis increasing to the right of the page origin and the positive Y axis increasing above the page origin.

Although it is initially placed in the lower left corner of the drawing, you can position the page origin anywhere on the drawing.

Geometric Coordinate Planes (GCPs)

DynaCADD's three dimensional coordinate planes are graphically displayed as three arrows pointing along the axes +X, +Y and +Z (positive X, Y and Z). The arrow pointing along the current X axis has an X marked on the tip.

DynaCADD contains seven predefined GCPs, with the option for you to create as many as nine more custom GCPs. The predefined coordinate systems are:

- Top
- Front
- Right
- Bottom
- Rear
- Left
- Isometric (Iso)

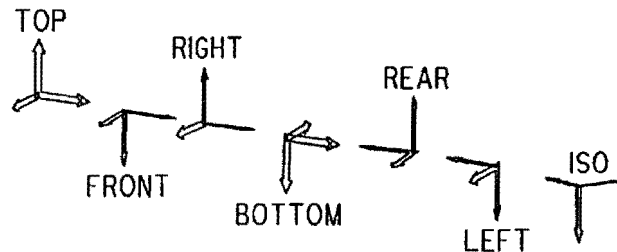


fig. (8) The Predefined GCPs

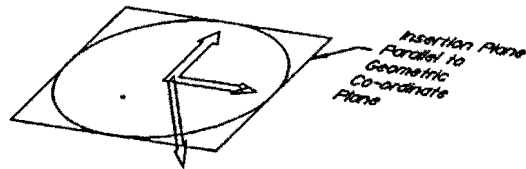


fig. (9) GCP Plane

GCPs are more than just a coordinate system defining the X, Y and Z axes; you can use them to specify on which plane you are working. The current plane in DynaCADD is an imaginary sheet positioned normal (perpendicular) to the Z axis at locations X0, Y0, Z0.

For example, consider inserting a circle into space. Although DynaCADD knows the origin and radius of the circle, there is no way of knowing the circle's 3-D orientation. To overcome this problem, DynaCADD places the circle in such a way that the circle lies parallel to the current X-Y axis.

By changing GCPs when you insert arcs, you can insert the arcs on different sides of a cube without having to exit the command.

GCPs are a very powerful feature that require patience and perseverance to use properly. Like any good tool, once learned, GCPs can provide you with the means of creating complex objects quickly and efficiently.

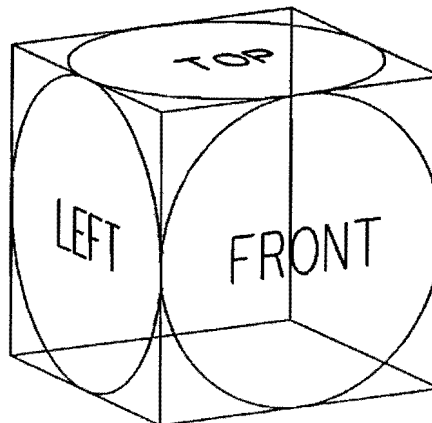
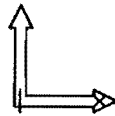


fig. (10) Multiple Planes

GCP Aligned with View



GCP Skewed to View

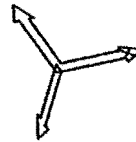


fig. (II) Aligned and Non-Aligned GCP

Free Digitizing

When a view and the current GCP are aligned, the positive X axis points to the right and the positive Y axis points up. The Z axis is not visible.

Digitizing is a means of specifying locations by positioning the crosshair and clicking the left mouse button. When digitizing, DynaCADD places the location on the current plane. However, if you select a location in a view where the GCP is not aligned with that view, DynaCADD has no way of knowing where, along the imaginary line travelling into the screen, you wish to place the location. DynaCADD can only guess as to where you wanted the point to be. You are informed of this uncertainty by the display:

“Warning: GCP Skewed to View”

If unexpected results occur in 3-D mode, check for this message on the command history screen.

To overcome this problem, you must ensure that the view and the GCP are aligned. You can tell this by looking at the GCP symbol. If the X-Y arrows are flat against the screen, then the GCP and the view are aligned. To fix an unaligned GCP and view, execute the command VIEWGCP and select the current GCP. This can save considerable frustration.

Free digitizing does not occur when entity snap is turned on. Entity snap makes the selection of a location absolute and is covered in detail in Chapter 5.

Layers

Each drawing in DynaCADD is actually made up of separate sheets of "clear paper", one on top of the other, each referred to as a layer. When you create a drawing using DynaCADD, you can specify on which of the individual sheets you would like the new entities placed. You also have the ability to turn on or off any combination of layers. One common use of layers is to have all groups of related entities on their own layer. By turning the individual layers on or off, it is possible to keep the visible drawing area uncluttered even when you create complex drawings.

In 3-D mode, layers also have depth perpendicular to the viewing plane.

In DynaCADD, you can use up to 256 layers, numbered 0 to 255. Layers can also be named. When the pointer is not over an icon, the layer number or name is displayed at the upper left corner of the drawing area.

To reiterate two important definitions:

Part: A part is a three dimensional database that defines an entity or a group of entities. Think of it as an actual object held within the memory of the computer. There are a large number of commands available, in order to manipulate the part in any way you desire.

Drawing: This is the actual page or sheet of paper on which you are working, with all zoom, view and image definitions. You can tell the computer the size of the paper and specify a scale factor for that page.

The User Interface ... Chapter 4

There are a several methods of communicating with and receiving information from DynaCADD, including:

Command and data entry:

- point and click command entry using the icons
- typing commands
- entering values into the value calculator
- the dynamic calculator
- selecting commands from the drop down menus

Information retrieval:

- displaying help and status messages
- displaying the command history and information window
- displaying online documentation
- echoing the command history to the printer

Screen and display control:

- drawing area display
- 2-D/3-D mode select

Corrections and restoration:

- the undo command
- command cancel
- deleting entities
- clear and restore

Command and Data Entry

Point and Click

The easiest and quickest method of entering commands is to use the mouse and click on the icons, or to select commands from the menus (see below and Chapter 5).

The User Interface

On the left side of the screen and across the top of the drawing area, small icons represent commands graphically. Each icon represents a command or, more commonly, part of a command sequence that you build by successively clicking on the icons in the correct order. The command name associated with each icon appears at the top left of the drawing area when you move the pointer over an icon.

Most commands require you to click more than one icon, sometimes as many as four or five icons must be selected: primary command, sub-command and modifiers or flags. With some commands, you must pause and enter specific information, such as angle, distance, or select a location or entity, before continuing.

Due to the large number of commands available, it is impossible to display all of the commands at the same time, so a tree structure is implemented in which related commands and modifiers only appear after a higher level command is selected. When you click with the pointer over an appropriate icon, the icon pad below it is drawn with the respective sub-commands.

DynaCADD uses four groups of icons on the left of the screen to display the available commands and modifiers. They display commands in a vertical hierarchy, from the top down. Clicking on the top level icon causes all secondary commands, or sub-commands, to appear in the icon pad directly below the top level icons. The main, or primary, command icons, reading left to right, top to bottom, represent these commands:

INSERT	CURVE	DIMENSION	TRANSFORM
EDITVIEW	CREATE	OUTPUT	INFORM
EDITLAYER	(unused)	MODE TOGGLE	UNDO

Additional appropriate sub-commands, modifiers or flags appear in the third icon pad, depending on which secondary level command you select. A flag is an optional modifier, such as "LINANG", line angle. In the reference section flags are described with the primary command and where applicable to a specific command noted as "(flag)". Other modifiers may be mandatory, such as entering a value for a circle radius when using the INSERT CIRCLE CIRRAD command or entering a text string for an INSERT TEXT command. These modifiers are noted in the command description with the modifier's name or abbreviation such as "(string)" or "(rad)".

The bottom icon pad is reserved for displaying the various entity and location selection commands, when applicable. These are all described in Chapter 6.

Most commands have defaults; icons are highlighted to indicate the default sub-commands and modifiers for any default state.

Many commands require exact information to execute: distance, angle, a selected entity, and so on. Whenever more information is required (not optional), a message appears at the bottom of the text screen, telling you what is needed. If one or more locations or existing entities must be chosen, do so by clicking the pointer at the appropriate locations, then press the right mouse button to indicate your selection is complete.

Whatever you select in the icon pads is echoed into the command history and can be examined by pressing [Tab] (see below).

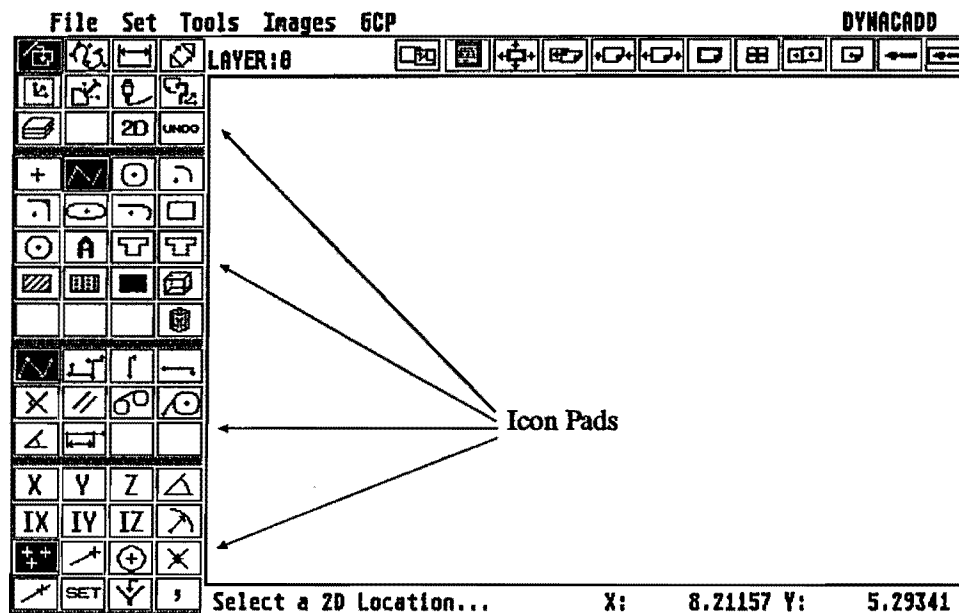


fig.(12) Icon Pads

Keyboard Command Entry

As an alternative to using the mouse, you can enter commands on the keyboard. DynaCADD interprets your commands as you type.

Associated with each icon is a command name, or a part of a command, that appears at the top of the screen when you move the pointer over an icon. This is the name you type to enact that particular command. Each command must be entered fully, starting with the name of the primary icon, then adding any names for modifiers, flags, etc. For example:

INSERT LINE LINELOC

is the command for inserting a line at the locations specified when you click the left mouse button.

The command syntax is very simple:

verb + noun + (flags and/or additional modifiers)

plus any data entry, entity or location selection as required.

For example:

(verb)	(noun)	(modifier)
INSERT	LINE	LINELOC
(flag)		(location modifier)
LINANG (angle)		loc #1, loc #2

Locations can be entered as absolute or relative (incremental) values by pressing [Enter]. For absolute coordinates, enter the X, Y (and Z in 3-D) values, separated by a coma. For example:

10,20	X = 10, Y = 20, Z = unchanged
,20	X = unchanged, Y = 20, Z = unchanged
10,,30	X = 10, Y = unchanged, Z = 30
,,30	X = unchanged, Y = unchanged, Z = 30

Input Absolute [X], [Y], [Z] Co-ordinates

10,,30|_____

OK

fig. (13) Absolute Coordinates Dialog

Relative values are measured as distances from the current (last) entity or location coordinates. Prefix the coordinate list with "i" to indicate incremental coordinates. For example:

- | | |
|-------------|---|
| i10,20 | X = increment 10 units over from the current
X location, Y = 20 from the current Y,
Z = unchanged |
| i,20 | X = unchanged, Y = 20 from the current Y,
Z = unchanged |
| i-10,-20,30 | X = increment -10 units over from the current X
location, Y = -20 from the current Y, Z = 30 from
the current Z |

Click OK or press [Return].

After you enter the final line location, press [;] (equivalent to clicking the right mouse button) to indicate entity selection is complete. For example:

1. Type: INS [Space] LIN [Space] for the command. LINELOC, for line location, is the default so it need not be typed.
2. Press [Return] to display the coordinates dialog.
3. Type: 10,10 [Return] to enter an X, Y location.
4. To enter another location, which adds another line segment to the series, press [Return].
5. Type: i10,10 [Return] to enter a second location relative to the first. The crosshair now moves to this location.
6. Repeat steps 5 through 7 with appropriate coordinates, until all required locations are entered.
7. Press [;] to end the location selection process.

Data entry such as layer name, number or copies, etc., and entity or location selection can occur anywhere, depending on the command. For data entry, a dialog or calculator pops up when necessary. An appropriate message appears at the bottom of the text screen when entity or location selection is required.

The User Interface

After typing in the first few characters of each part of the command, press the space bar. This enters the command into the interpreter and tells DynaCADD to select the icon represented by the typed command. You do not need to enter the full command name, only enough of the command to identify it; DynaCADD performs a comparison on whatever has been typed and selects the first command name that matches your entry. For example:

INS [Space] LIN [Space] ORT [Space]

is the same as:

INSERT LINE ORTHO

You can click icons and type commands interchangeably, or press the function keys at any appropriate place during comand entry.

Commands are stored in a command history. To see the command history, or any other text and information, press [Tab].

Function Key Macros

The ten function keys, labelled [F1] to [F10], are preset with some of the most commonly used commands, sub-commands or modifiers. When you press the key, the text is inserted into the command line processor and executed exactly as if you typed it or clicked on the icon. The keys and their associated text are:

[F1]	LOC	Location
[F2]	END	End
[F3]	ORG	Origin
[F4]	INT	Intersection
[F5]	BYDIG	By digitizing
[F6]	BYWIN	By window
[F7]	BYPWIN	By polywindow
[F8]	DZALL	Draw zoom all
[F9]	DZWINDOW	Draw zoom window
[F10]	REPAINT	Redraw the screen

These commands are discussed in more detail in the command section. The preset function key definitions cannot be changed.

Value Calculator

Whenever DynaCADD requires you to enter a numeric value, a pop up scientific calculator is displayed. This operates much the same as a hand held calculator. To activate a function, or enter a number, position the pointer over the appropriate symbol and click. You can also enter the values through the keyboard. Results are displayed in the calculator window, at the top.

Like a real calculator, you must click [=] to get a calculated result. With negative numbers, you must click [-] before you enter the value. To see the negative sign, click [=].

The symbols on the calculator are mostly self explanatory. Here are a few notes on the actual operation:

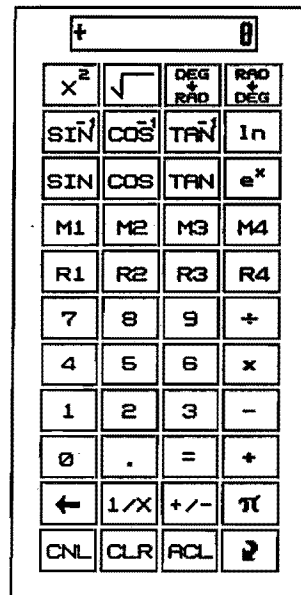


fig. (14) Value Calculator

- Precedence of operators is strictly in order of entry.
- The keys M1 to M4 represent four memory storage buffers. They store the value currently in the calculator window. These memories will remain intact throughout the DynaCADD session.
- The keys R1 to R4 recall the values stored in memory locations M1 to M4, respectively.
- Pressing [,] on the keyboard or calculator is equivalent to pressing [Return], followed by [,]. This key is a shortcut when entering and activating location coordinates (see Location Selection, Chapter 6).
- The CLR button clears the current value in the window.
- The ACL button clears all previous operations.
- The CNL button (or [Undo]) cancels and closes the calculator.

The User Interface

The Dynamic Calculator

Unlike the value calculator, the dynamic calculator does not accept numeric input. It is designed to let you change 3-D view rotation, scale and zoom levels interactively by clicking arrows that represent increments and see the effect as you work.

The three parts of the dynamic calculator are:

- Rotation
- Scroll
- Zoom

Their functions are described in more detail in Chapter 11.

Drop Down Menus

The drop down menus provide access to global commands, utilities and functions that you need to have available at all times.

To access the menu bar, move the pointer to the top of the screen over a menu title. The appropriate menu will drop down, revealing the options available under that particular title. To execute a command, move the pointer over the option and click. If a menu option is shown in a lighter text, that entry is disabled and currently unavailable for selection. To close a menu without making a selection, move the pointer away from the menu items and click the mouse button.

The actual function of each menu item is covered in Chapter 5.

Information Retrieval

Help and Status Messages

Due to the large number of available commands, it is very difficult for all but the most dedicated user to remember the name and function of each of the over 300 icons. To overcome this problem, DynaCADD can optionally display the command name and a short help message every time the pointer passes over a command icon. You can also type this command name directly (see Keyboard Command Entry, above).

The command name is displayed at the upper left corner of the drawing area when the pointer is positioned over an icon. Otherwise this line displays the layer name or number. A brief help message is also displayed in the bottom right hand corner of the screen, invaluable to the novice or occasional user. If the help message status is turned off, the command name and help message are not shown.



Command History

The command history window is a text display that shows the command history of the current session, as well as various drawing data and the results of information commands such as measure area. Press [Tab] to view it. Press [Tab] again to return to the design session. Clicking the SHOWCOM icon is the equivalent to pressing [Tab].

The command history window is a continuous scrolling display. Text that disappears from the window is not lost, it is merely hidden and can be viewed by moving the slider bars or clicking on the arrows at the top and bottom of the window. The command history does use up memory and during a long design session, a considerable amount may be consumed. Press [shift-Delete] to clear the command history and free up memory.

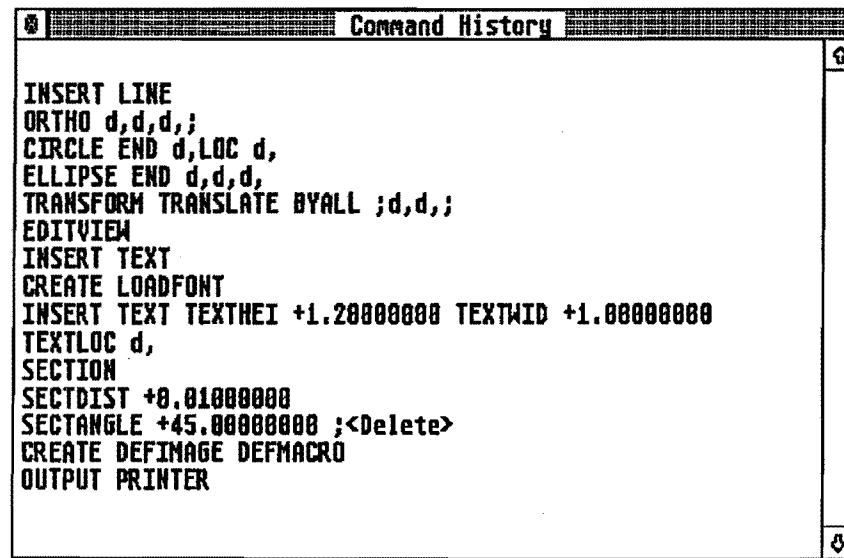


fig. (15) Command History

Online Documentation

If you find the help and status messages are insufficient, press [Help] to activate the full online documentation feature. A window appears, with the appropriate text explaining the currently active command displayed. To pause and adjust the scrolling speed use:

[Ctrl-S]	Stop Scrolling
[Ctrl-Q]	Restart Scrolling
[Ctrl]	Slow Scrolling
[Esc]	Cancel online documentation
[Alt]	Fast scrolling

When all of the documentation describing the use of the active command has been displayed, an "End of File" message appears at the bottom of the window. Press any key to return to the design session.

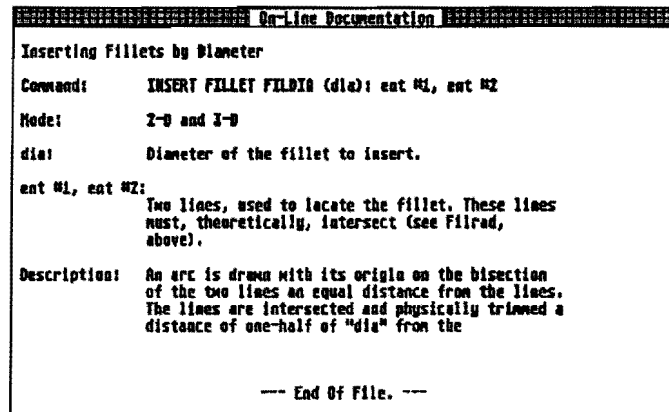


fig. (16) Online Documentation

Printer Echo

You can echo the contents of the command history and text window to the printer by selecting this option from the Tools menu. This is very useful when generating reports through the Inform commands (see Chapter 14).

Screen and Display Control

Managing the Drawing Area



There are many transformation (change) commands in DynaCADD that are affected by this command (TRANSLATE, ROTATE, etc.). When TRANCOPY is on, any transformation commands leave the original entities being acted upon untouched, and create a copy before applying the particular transformation. If TRANCOPY is off, any transformation commands act upon the original entities selected.



To view the command history and text display window, click on the SHOWCOM icon or press [Tab]. The slider bars and arrow buttons are for scrolling the display in the appropriate direction, up or down. To exit the command history, click the close button at the upper left of the window. You can also press [Tab] to display and exit the command history window.

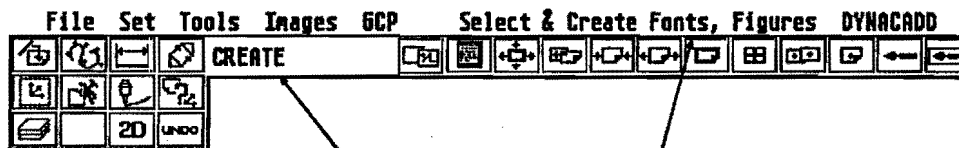


fig. (17) Command Name and Help Messages

Drawing Modifiers

There are nine drawing modifier icons to control which portion of the page is contained within the current drawing area. These commands do not affect currently active command sequences.

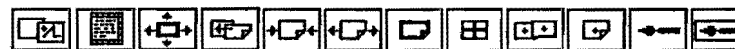


fig. (18) Drawing Modifiers

The User Interface



View Drawing Extents (DZEXTENTS)

Show the full database in the drawing area. The display is sized so that all visible entities are displayed. This works best in 2-D mode; in 3-D mode it also includes all of the active view windows. For 3-D drawing extents, it is better to use the VIEWZEXT icon from the EDITVIEW icon pad.



Zoom Last (DZLAST)

This command displays the previous drawing extent.



Zoom In (DZIN)

This command requires the selection of a 2-D location. Once you select the location, the drawing area is zoomed in (magnified) by a factor of two. The center of the drawing area is the selected location.



Zoom Out (DZOUT)

This command requires the selection of a 2-D location. Once you select the location, the drawing area is zoomed out (reduced) by a factor of two. The center of the drawing area is the selected location.



Zoom All (DZALL) [F8]

This command zooms the full drawing into the drawing area. As with most of the drawing modifier commands, the previously active command continues uninterrupted.



Zoom Window (DZWINDOW) [F9]

This command requires the selection of a window. Once you select the two locations that define the window (diagonally opposite corners), the portion of the drawing defined by the window is zoomed in and displayed.



Scroll Drawing (SCROLL)

This command requires the selection of two 2-D locations. After you select the first location, a dotted rectangle representing the current drawing area is displayed and moves with the full screen crosshair. After you select the second location, the drawing moves from the first location to the second, at the same zoom factor.

Note: To exit the Zoom In, Zoom Out, Zoom Window and Scroll commands, click the right mouse button or press [;].



Center Drawing (CENTER)

This command requires the selection of a 2-D location. Once you select it, the drawing is centered on the location, at the current zoom factor.



Repaint (REPAINT) [F10]

Sometimes when you edit a drawing, particularly when you delete entities, the graphics are partially erased. This is simply a screen effect; the entities are not actually changed. Select REPAINT to refresh all entities visible in the current drawing area.



Regenerate Graphics (REGEN)

Use this command to regenerate the entire drawing and all existing views from the information contained in the database. If, for any reason, the graphics displayed do not accurately represent the database, use this command to restore the drawing.



Mode Select (INIT2D/INIT3D)



There are two basic modes of operation: 2-D and 3-D. Entities created in one mode are not available for modification in the other, although all entities are visible in either mode.

To switch modes between 2-D and 3-D at any time during the design process, click on the 2-D/3-D mode toggle icon on the main icon pad. The currently active command is cancelled and the mode is toggled.

Note: You must open a view in 3-D mode in order to manipulate entities in 3-D. See Chapter 11.



Corrections and Restoration

With a few exceptions, you can undo the last major command by clicking on the Undo icon in the main icon pad, or by pressing [Undo]. After verifying your request, DynaCADD restores the drawing to its status before the last command was executed. Any currently active command is cancelled.

If you do not want to execute a command in progress, you have several options:

- select another command or click another icon
- click Cancel in a dialog, file selector or calculator

When inserting entities, press [Delete] to erase the last entity drawn. If the entity is a "chained" entity, such as continuous lines or filled solids, [Delete] erases the last entity, one at a time, in reverse order. [Delete] also takes back multiple locations in the reverse order in which they were selected.

You can clear the entire database by selecting Clear from the File menu, or restore the database to the most recent save by selecting Restore. These are described in Chapter 5. Use these commands with caution, since they cannot be reversed with UNDO.

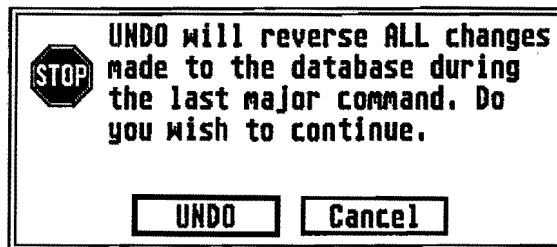


fig. (19) UNDO Dialog

Drop Down Menus ... Chapter 5

To aid in the drawing and design process, numerous commands are grouped under the appropriate headings in the drop down menus. In addition to the menus, there are additional screen modifier icons that are grouped together above the drawing area, described at the end of this chapter.

On the menu bar, across the top of the screen, are six menu titles:

File Set Tools Images GCP DYNACADD

Under each title are numerous related entries called options. To access a menu, move the pointer up to the appropriate title. The menu drops down onto the screen, displaying its list of options. Move the pointer onto the required entry and click the left button. DynaCADD now executes the command indicated by the selected menu option. To close a menu without making a selection, move the pointer away from the menu items and click the mouse button.

Some of the menu entries have an inverse diamond symbol and a letter beside them. This means you can activate that command by pressing [Alt] plus the indicated letter. For example [Alt-X] brings up the File Transfer dialog.

DynaCADD Menu

DynaCADD® Information Dialog

Under the DYNACADD menu there is an Information entry. Clicking on this item brings up a dialog with the following information:

- The DynaCADD version number.
- The name and address of the licensee.
- The serial number.
- The system's processor and math co-processor types.

You will need this information whenever you call Technical Support with questions on DynaCADD or Customer Service for an update.

Click anywhere in the dialog to erase it.

The Drop Down Menus

File Menu

The File menu contains all the various storage and retrieval commands, in addition to some auxiliary commands related to database manipulation.

When DynaCADD completes the Save Part, Save Default, Save Macro, Merge Drawing, Clear, Restore and Database Sort commands, it clears the icon pads and cancels any previously active command. Only Memory Status and Quit do not do this.

Save Part [Alt-S]: This command saves both the current part and drawing to the filenames specified at the time they were activated. The entire session is saved, with all parameters such as view definitions, images, and coordinate systems being saved for later recall. When a drawing is loaded later on, you are returned to the state of the most recent save.

We recommend that you use this command at intervals no greater than 30 minutes apart. In the event of a power failure, this practice minimizes the amount of work lost. In the event of a disk error, you are notified of the problem. We also recommend that you keep backups of your work on floppies, in case anything goes wrong.

Save Part As: This is the same as Save Part, except that you can specify a different filename. DynaCADD adds the correct extensions to the filenames, if you forget to enter them.

Save Default: Whenever a new drawing has been activated, DynaCADD needs to know how to initially set the drawing parameters:

- page width, height and scale
- GCP, view and image definitions
- dimension status
- tool values
- layer names
- line style definitions

In order not to reset these parameters every time a new drawing is created, you can determine the initial setup by a default drawing that you create. To save the default drawing parameters:

1. Set all parameters to their desired state.
2. Use the Save Default command.

Once a default drawing has been saved, every new drawing activated is initially set to the predefined defaults.

Save Macro: You can assign multiple keystrokes to a keystroke combination such as [Ctrl-A]. The Save Macro command saves these macro definitions in a file. You cannot change the Function key macros. For more information on defining and using macros, see Chapter 12. For information on listing macros, see Chapter 14.

Merge Drawing [Alt-M]: This command allows you to combine other drawings with the current drawing. When combining drawings, only 2-D entities are loaded. The drawing parameters of the current drawing, such as page width and height, are left unchanged. When you select this command, the File Selector dialog appears. You can load the drawing to merge from any drive and directory, providing it is a valid DynaCADD file.

File Transfer [Alt-X]: This is DynaCADD's file conversion dialog. You can import the following files:

- DXF (AutoCAD)
- DEF (DynaCADD)

and export these file formats:

- DXF (AutoCAD)
- DEF (DynaCADD)

See Appendix A for further information on DXF and DEF file transfers.

The Drop Down Menus

- Clear:** This command clears the database (all entities) from memory. All loaded font definitions are also cleared, freeing all memory. **Warning:** You cannot UNDO this command. Any changes to the current part and drawing are irretrievably lost.
- Restore:** This command reloads the current part and drawing from disk. This is normally used after some design error has made it impractical to try and rebuild the part using the normal editing commands. Save and Restore are also a part of rebuilding contiguous memory blocks (see Memory Status, next page following). **Warning:** You cannot UNDO this command. Any changes to the current part and drawing are irretrievably lost.
- Database Sort:** During the design process, a certain amount of “garbage” is built up in the database as a result of deleting and modifying entities. This command cleans up all the garbage. It also releases all inactive memory not being used by the database. It is a good idea to use this command at regular intervals. This command also displays the Memory Status dialog after it has completed.

Memory Status

This command displays the Memory Status dialog telling you the amounts used and free of expanded and system memory. Click anywhere inside the dialog to remove it.

MEMORY STATUS	
EXPANDED	SYSTEM
Total : 2292640	Total : 211568
Used : 199140	Used : 79968
Free : 2093500 (91%)	Free : 131600 (62%)

fig. (20) Memory Status Dialog

- Expanded:** This panel shows the total amount of expanded, or EMS, memory that DynaCADD has for data storage. The amount currently used by DynaCADD for screen buffer, font and drawing storage, and the amount of expanded memory currently free along with a percentage.
- System:** This panel shows the amount of conventional, or system, memory that DynaCADD has available to it. The amount used by DynaCADD, and the amount of system memory currently free along with a percentage. For proper and speedy operation during printing and file transfers, the free amount should be at least 64000 bytes and not go below 32000 bytes.

The Drop Down Menus

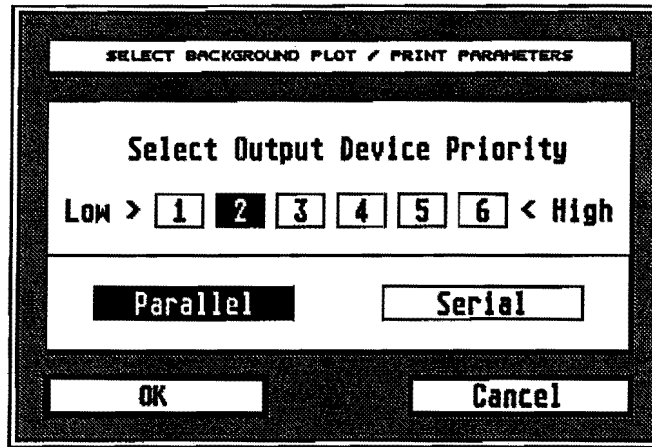


fig. (21) Background Output Dialog

Background Output:

Background output allows you to continue to work in DynaCADD concurrently with a drawing's output on your plotter or printer.

The dialog lets you select the priority of processing time used for plotting. A low priority means the output takes longer, but you will not notice much, if any, slow down in DynaCADD. A high priority means faster output, but some slow down in DynaCADD.

You must also select the correct output port, serial or parallel.

When you click OK, enter a filename in the File Selector dialog. This must be the name of a valid .PFL file, previously saved to disk (see Chapter 12). The drawing on disk is redirected to the plotter or printer while you continue to work in DynaCADD.

To cancel the output, click on Background Plot again.

Plotting and printing are described in detail in Chapter 12 and in Appendix D.

Quit [Alt-Q]

This command exits the current session. After you select Quit, a dialog appears offering three options:

- Quit:** Click here to exit the current session, without saving any of the changes that may have occurred since the last time the part and drawing were saved.
- Save:** Click here to save the part and drawing before exiting the current session. Before you exit, a file is saved to disk informing DynaCADD which part and drawing to load the next time the program is run. The normal procedure of part and drawing setup at the System level is bypassed, and at the start of the next session, you are given the opportunity to automatically restore to the current status.
- Cancel:** Click here to cancel the Quit command and return to the current session.

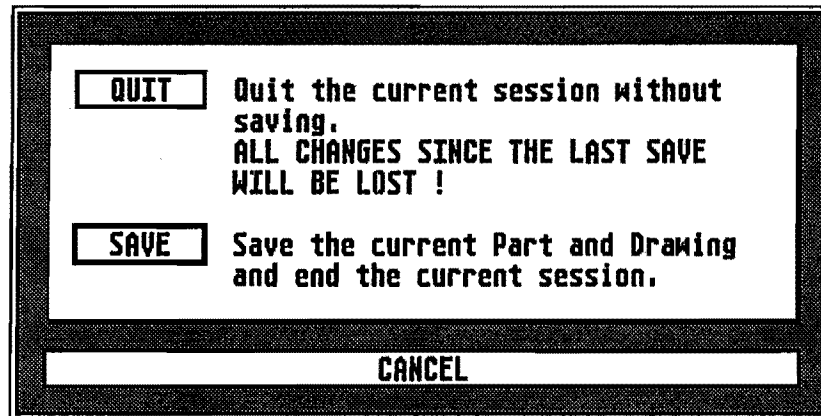


fig. (22) Quit Dialog

The Drop Down Menus

Set Menu

The Set menu contains ten entries, seven of which control the parameters for the following functions:

Set	
Drawing...	⌘N
Dimension...	⌘D

Track/Grid/Axis...	

Pen Indexes...	⌘P
Pen Style...	

Layers...	⌘L
Hatch Pattern...	

Preferences...	
Color Palette...	
Paper Color...	⌘Z

Drawing [Alt-N]:

This command displays the System level dialog. You can load a new part or drawing, change units and scale here. For a complete discussion on the System level, see Chapter 2.

Dimension [Alt-D]:

This is the dimension parameter definition dialog. Such elements as the arrowhead type, text format and precision are established here. For a complete discussion of the dimensioning capabilities of DynaCADD, see Chapter 9.

Track/Grid/ Axis:

Select this command to edit the parameters associated with the dynamic tracking, grid and axis drawing aids. After you select the desired values, click on the OK button to install the changes and return to the previously active command.

Pen Indexes [Alt-P]:

This is where you determine the current pen parameters: pen number, color, weight and line style.

Pen Style:

Select this to create or edit, and name, line style definitions for use with the pens (see above).

Layers [Alt-L]:

This command displays the Layer Attributes dialog where layer information can be manipulated directly. Layer attributes govern layer names, color, pen weight and style and layer status. For a complete discussion of layers, see Chapter 15.

Hatch Pattern: This is where you choose the current pattern for use in all subsequent hatch commands.

Tracking

This refers to the ability of DynaCADD to display the current 2-D position of the full screen crosshair on the drawing. The coordinates are displayed in the top right hand corner of the screen. In the Tool dialog, you can specify the format of the displayed coordinates:



fig. (23) Tool Parameters Dialog

- Absolute Cartesian:** The coordinates displayed are the absolute X and Y coordinate using the page origin as 0, 0.
- Relative Cartesian:** The coordinates displayed are the X and Y coordinate values, using the current location of the current command as the origin 0, 0.
- Absolute Polar:** This displays the angle and radius (distance) of the cross hair, relative to the page origin.
- Relative Polar:** This displays the angle and radius (distance) of the cross hair, relative to current location of the current command.

The Drop Down Menus

Grid Values

Here you define the grid spacing, in the current units. There are two types of grid markings: major and minor. Major marks appear as darker points. You can, for example, mark feet as major marks, and inches as minor marks.

Entities and locations are automatically snapped to the closest grid point when the grid is on. To edit a grid increment:

1. Click the pointer on the Grid X button. This is the X axis increment.
2. Press [Esc] to clear the current value.
3. Type the desired X increment.
4. Click the pointer on the Grid Y button. This is the Y axis increment. It does not have to be the same as the X axis increment.
5. Press [Esc] to clear the current value.
6. Type the desired Y increment.

To set the major marking values, click on the Major button and then click on the X and Y buttons and enter the increment. The major markers are not measurements like the grid, but integers which divide the grid by the numbers entered. For example, to have a major mark every fourth grid point, enter 4 in the X and Y buttons.

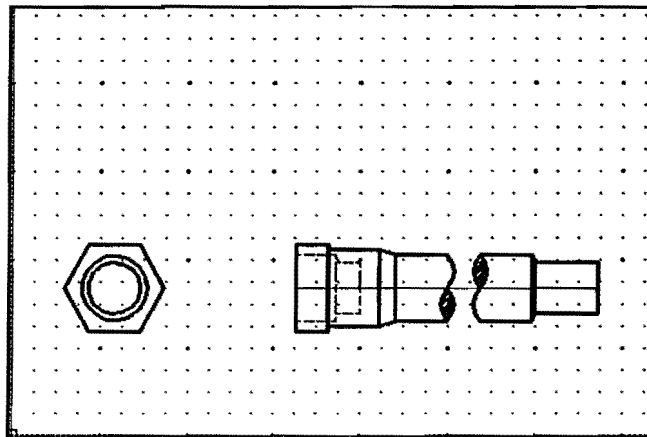


fig. (24) Grid On

Axis Values

The axis provides a reference drawing outline, perpendicular to the screen. As with the grid, you can define both the X and Y increment, major and minor markings, in both axes. The axis appears as lines, rather than points. There is no snap associated with the axis. To edit the axis increment, follow the steps outlined in grid, above.

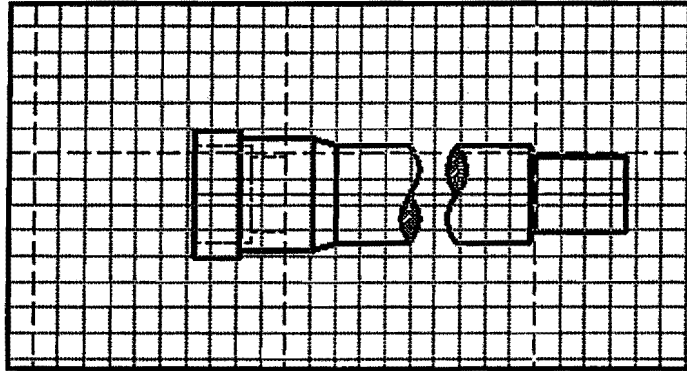


fig. (25) Axis On

Pen Indexes [Alt-P]

This dialog is where you determine the current pen parameters: pen number, color, weight, and line style.

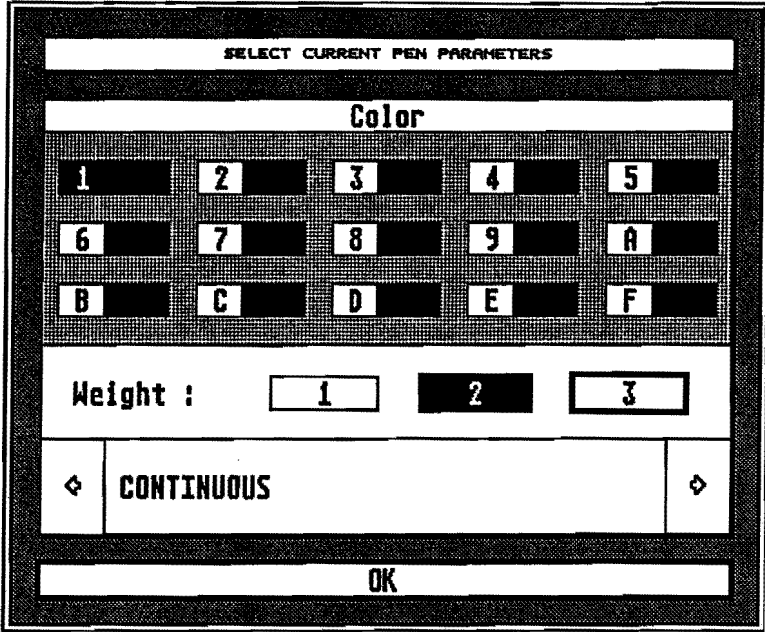
There are 15 pens, numbered 1 to 9, and A to E. Each of these relates to a pen on your plotter. Select the pen by clicking on the label. Colors associated with each pen depend on your plotter. The current pen is highlighted.

There are three line weights available, numbered 1 to 3, in order of increasing thickness. Click the thickness value for each pen.

The line style is represented by a name such as "CONTINUOUS", "DASHED", "CENTER", and so on. These names are either system assigned or those which you enter yourself when you define the line style (see Pen Style, below). Each entity can have its own line style. Click on the right and left arrows to scroll through the list of line style names.

All entities and dimensions are drawn in the current pen color, weight and style. The default line style is continuous.

The Drop Down Menus



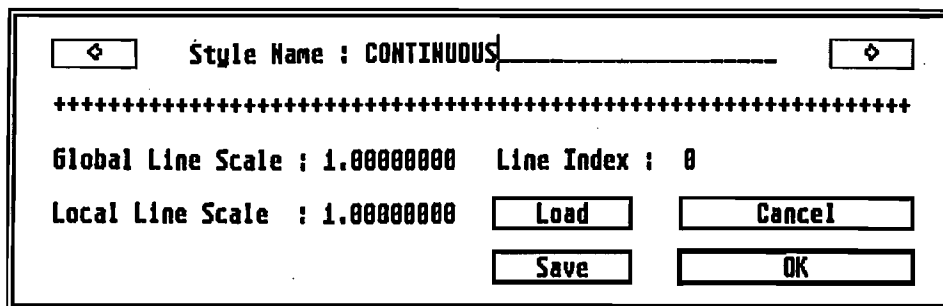
The image shows a 'Pen Parameters Dialog' box. At the top is a title bar that says 'SELECT CURRENT PEN PARAMETERS'. Below this is a section labeled 'Color' which contains a 3x5 grid of buttons labeled 1 through F. Below the color grid is a 'Weight' section with a label 'Weight :' followed by three buttons labeled 1, 2, and 3. Below the weight section is a large button labeled 'CONTINUOUS' with diamond-shaped icons on either side. At the bottom of the dialog is an 'OK' button.

fig. (26) Pen Parameters Dialog

Pen Style

This is where you create or edit, and name line (pen) style definitions for use with the pens (see above). There are forty nine user definable line styles, numbered 1 to 49. Do not edit line styles numbered 50 and up; these are reserved for system use and file translations. Also, do not change any styles with a "SYS_" prefix in their name. These are used by the system for many commands.

The top line is the line style name. Below it is an ASCII representation of the line. Below that are the global and local scales.



The dialog box has a title bar with a diamond icon on the left and a close button (X) on the right. The main content area contains the following elements:

- Style Name:** A text field containing "CONTINUOUS" with a cursor at the end. To the right of the text field is a small diamond icon.
- Definition Line:** A line of 64 asterisks (*****).
- Global Line Scale:** A text field containing "1.00000000".
- Line Index:** A text field containing "0".
- Local Line Scale:** A text field containing "1.00000000".
- Buttons:** Four buttons are arranged in a 2x2 grid: "Load", "Cancel", "Save", and "OK".

fig. (27) Pen Style Definition Dialog

To create a line style:

1. Select either an unused style (named "undefined"), or one of the existing styles. Click on the right and left arrows to find a style.
2. Press [Esc] to clear the name and enter your own style name.
3. Click the cursor on the definition line below the name. This 64 character line is where you create the line itself by typing any character (see below).
4. Set the scale (if necessary).
5. Click OK.

Each character in a line definition represents a solid part of the line; a space represents a blank part. DynaCADD treats each character and space as a ratio of the whole. Remember that the beginning and end of the definition are joined when the line is drawn on the screen. For example, if you type:

**** ****

this creates a line where two thirds is solid and one third blank, like this:

And if instead you type:

*** ** ***

The Drop Down Menus

you end up with a line divided into even dashes:

— — — — —

Since each part is examined as part of the whole, this:

is the same as:

**

because each segment is the same proportion of the entire length (one third). You should design your lines so that each end has a solid, so that blanks do not appear in corners or angles.

Global Scale: This changes the scale of all lines in the drawing, immediately. Think of it as the “zoom” factor for the line. The size of the solid and blank segments of all lines on the screen are sized according to this value. The larger the value, the larger the segments.

Local Scale: This changes the scale of only those lines with the displayed name. If the local scale is changed, all lines drawn with this style automatically update.

Load/Save: You can load and save line style definitions as .STY files. The File Selector dialog appears when you select one of these options. Note that line style definitions are also saved with each drawing.

Layers [Alt-L]

This dialog displays all the information associated to several layers. There are two methods of scrolling through the layers. The small arrows scroll one page at a time. The larger arrows scroll in three page jumps. To modify a layer attribute, simply select the attribute and the dialog associated with that attribute will appear. The attributes govern the names and pen color, weight and style of a layer. This is the only means of changing a layer's pen color, pen weight and pen style. There are three status flags that can also be selected or deselected. These flags display whether or not the layer is visible, active or locked.

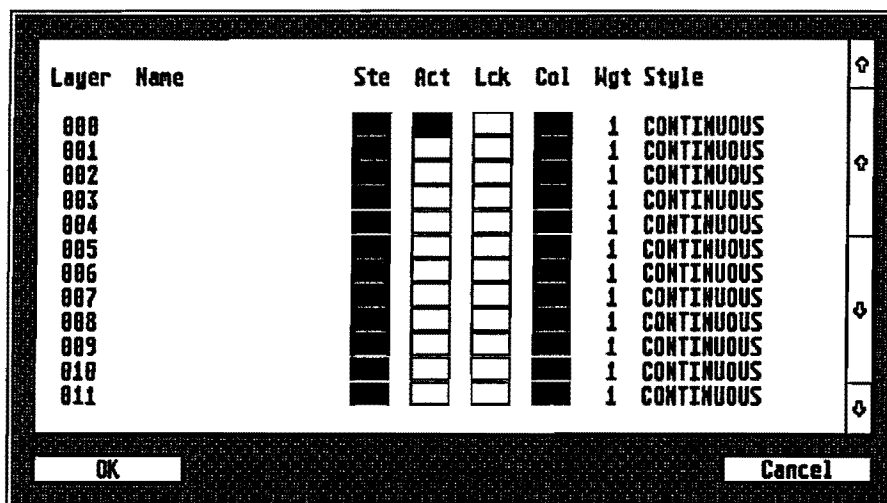


fig. (28) Layers Dialog

The column titles and descriptions are as follows:

Layer	Layer Number	This can not be changed.
Name	Layer Name	Change this with a Layer Name dialog.
Ste	Layer State	This is the visible status flag.
Act	Layer Active	Only one layer can be active at one time.
Lck	Layer Locked	This is the locked status flag.
Col	Layer Color	Change this with a Pen Color dialog.
Wgt	Layer Weight	Change this with a Pen Weight dialog.
Style	Layer Style	Change this with a Pen Style dialog.

The Drop Down Menus

Hatch Pattern

The current pattern used in all hatch commands is shown in the window. Click on the right and left arrows to scroll the display of available hatch patterns.

The hatch patterns are derived from a font file called "HATCH.FNT". These patterns can be modified in the Vector Font Editor.

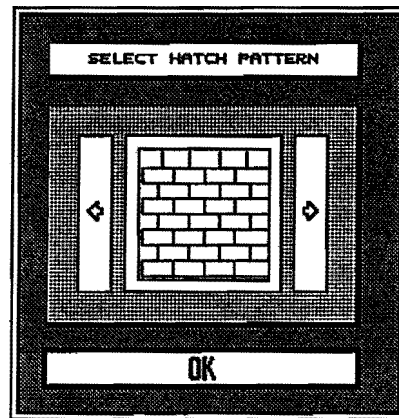


fig. (29) Hatch Pattern Dialog

Preferences

The Preferences dialog allows the choice of which serial (COM) port DynaCADD will use for its serial output to plotters or printers, and whether or not the system shall beep audibly on errors.

Serial Port: Any one of four serial ports, COM1, COM2, COM3 or COM4 can be chosen. Only those available on your system are selectable. Unavailable ports are grayed out. Be careful if your system has a serial mouse hooked up to it. If you choose the port that the mouse is plugged into, and click on OK, you will lose use of the mouse.

Two speeds, or baud rates, are available. 9600 and 19200. All plotters and printers support 9600 baud. Some high-end plotters also support 19200 baud. Initial testing and hookup for output should be attempted at 9600 baud.

After making your selections and clicking on OK, the COM port will be configured for 8 bits, no parity and 1 stop bit, along with the requested baud rate and hardware handshake protocol.

Note: DynaCADD runs with XON/XOFF constantly enabled. Calcomp plotters require RTS/CTS handshaking to be turned on, otherwise leave RTS/CTS off (unless your plotter documentation states otherwise).

Bell: If you work in a crowded environment, or find the beep of your machine annoying, you can shut it off by clicking on the respective button beside Bell.

To save settings so that they are retained from session to session, click on the Save Settings button.

The Drop Down Menus

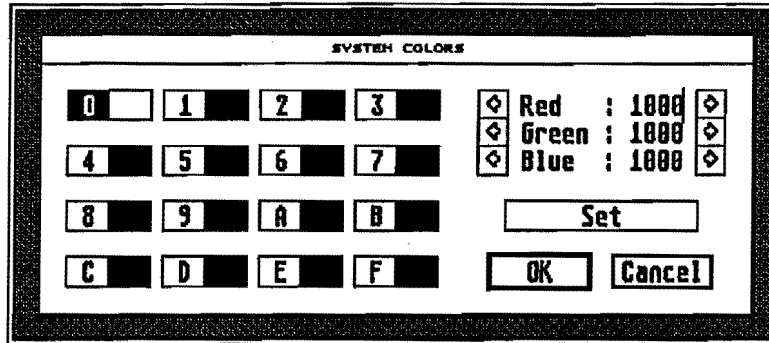


fig. (30) Color Palette Dialog

Color Palette

The Color Palette dialog enables you, on a color system, to change the color of the 15 available pens and the background.

To change a pen color, click on the letter beside the color you wish to modify. The level of the red, green and blue for that color will be displayed in number from 0 to a 1000. You can now click on the arrows beside the respective primary color to increase or decrease the value by tens. Hold down the shift key while clicking on the arrows to increase or decrease the value by ones. You can also directly enter in the amount from the keyboard and then click on the Set button.

If you wish to restore a color after it has been changed, but before OK is selected, press the shift key and click on the color to restore. To restore all colors, click on Cancel.

Color changes are saved with each drawing, this enables every drawing to have its own set of colors.

Paper Color [Alt-Z]

The Paper Color dialog enables you to change the color of the paper, or drawing area on the screen.

To change the color, click on the letter beside the color. The drawing area will change to that color and the drawing will repaint.

The paper color is saved with each drawing, this enables every drawing to have its own set of colors.

Tools Menu

The Tools menu contains various drawing aids and global editing parameters. Note that several tool commands act as toggle switches and, when turned on, are indicated by a check mark on the menu. You do not define the tool parameters here (see the Set menu, above), rather you simply indicate whether or not the tool is active.

Frame [Alt-F]: Click here to turn the frame display on or off. You can have as many as five drawing frames visible on a DynaCADD drawing. The first drawing frame borders the actual page, and is used as a visible representation of the drawing size. The additional four frames are the borders of any active 3-D views.

GCP [Alt-J]: Click here to turn the GCP display on or off. As discussed earlier, the Geometric Coordinate Plane (GCP) shows the current 3-D coordinate planes as a set of three arrows pointing along the positive X, Y and Z axes with its center at the part origin.

For a visible GCP, the following conditions are necessary:

- the selected mode is 3-D
- one or more views are active
- the GCP command has a checkmark

There are GCP related commands under the EDITVIEW and CREATE icons.

Drawing
Origin [Alt-O]: Click here to turn the 2-D page origin display on or off. This command is similar to GCP, but for 2-D mode. The drawing (page) origin appears as a small box centered on the current 0, 0 location of the page.

The Drop Down Menus

- Grid [Alt-G]:** Click here to turn on both the grid display and the grid snap. Unlike some CADD packages, the snap mode and grid visibility are not separate characteristics in DynaCADD. When the Grid is visible, snap mode is automatically activated. Snap causes all locations to appear at the nearest grid intersection. The major grid points appear darker than the minor grid points. Change the grid values in the Tool parameter dialog.
- Axis [Alt-A]:** Click here to turn on or off the display of axis reference lines. The axis lines are similar to the grid, but have no associate snap. The lines are spaced a definable 2-D distance from each other, starting at the page origin. The major axis lines appear darker than the minor axis lines. There is no snap associated with the axis. Change the axis values in the Tool parameter dialog.
- Graphics [Alt-Y]:** This displays the control points and direction of B-splines and Bézier curves.
- Tracking [Alt-T]:** This command displays the current 2-D location of the full screen cross hair, in any of four definable formats (see Dynamic Tracking, above). The X,Y coordinates appear at the right, above the drawing area.
- Help Messages [Alt-H]:** This turns on or off the display of command names and help messages at the top of the screen. Help messages are an invaluable aid to the novice or occasional user. When the pointer passes over an icon, a short help message describes that particular function.
- Close Curve [Alt-W]:** This command determines if the last point of a curve automatically joins with the first point (closes the curve) when a new curve is inserted.
- View Clipping [Alt-V]:** This restricts the display to the area defined by the current views. If View Clipping is turned on, only the portions of the 3-D part that are visible within the defined view areas are displayed. Otherwise the drawing page is used to limit the graphics.

- Rubber Band** When Rubber Band is on, a line is drawn from the
[Alt-R]: current location to the center of the full screen cross hair in such commands as INSERT LINE and POLYWINDOW. This rubber band line is only displayed in 2-D mode.
- Full Crosshairs** When Full Crosshairs is on, the crosshairs are large,
[Alt-C]: occupying the full screen. When Full Crosshairs is off, the crosshairs appearing on the screen are small in size.
- Text Boxed** When Text Boxed is on, all text entities are displayed
[Alt-B]: as a crossed box representing the physical extent of the text. If Text Boxed is off, the actual text string is displayed if the size of the text on the screen is large enough to read. This command is not applicable to any devices other than the screen. Output to plotters, printers and files always displays the actual text entity.
- The time it takes to regenerate the screen graphics is greatly reduced when Text Boxed is on.
- Printer Echo** During a design session, all commands and
[Alt-E]: parameter responses are echoed to the command history. Information commands also use this window for their output. Printer Echo also sends the output to a printer connected to the computer's parallel port.
- Screen Buffer:** If you have a system with more than 1 megabyte of memory, you can buffer the screen, so that it does not need to be regenerated after a dialog is displayed. This buffering speeds up screen redrawing considerably. However, if you need the additional memory the buffer uses, activate Screen Buffer. When Screen Buffer is off, the memory is available for your drawings.

The Drop Down Menus

Inheritance [Alt-I]:

As with TRANCOPY, inheritance is a transformation modifier. Whenever a transformation command is applied to an entity, inheritance controls the status of the new entity. If inheritance is on, the new entities receives the following attributes from the original entities:

- pen color and weight
- line style
- layer
- view status

If inheritance is off, the above characteristics are obtained from the current parameters (the selected attributes). Inheritance is normally left on.

Images Menu

An image is simply a stored definition of the zoom and scroll factors on the drawing.

Each image has its own name entered into the Images menu which can contain as many as sixteen images. You can retrieve them at any time. After an image is selected, the drawing area clears and redraws, using the extent supplied by the individual image definition. For a complete discussion of Images see Chapter 11.

GCP Menu

The GCP indicates the 3-D coordinate plane in a view. GCPs do not appear in 2-D mode. There are seven system defined Geometric Coordinate Planes (GCPs) available:

- Top
- Front
- Right
- Bottom
- Rear
- Left
- Isometric

There is also room for nine user defined GCPs (see Chapter 15). All available GCPs are shown in the GCP menu. The currently active GCP is preceded by a check mark in the menu. Creating and changing the GCP is described in Chapter 11.

To select a GCP for the active view, click on the name in the menu.

The Drop Down Menus

Location and Entity Selection ... Chapter 6

Virtually all entity insertion and modification (transform) commands require you to pass information to DynaCADD. The six basic types of information required are:

- location
- entity
- window
- view
- mode
- data and text

The various methods of selecting each type are covered in this chapter.

Note: There are default states for selection, indicated on the screen by the highlighted icons.

Location Selection

Whenever a location is required by DynaCADD, the bottom icon pad area automatically updates to display the location selection modifiers. With these modifiers, it is possible to specify locations using any or all of the following:

- | | |
|----------------------------------|----------------------|
| • pointing | LOC |
| • absolute Cartesian coordinates | X, Y, Z |
| • relative Cartesian coordinates | IX, IY, IZ |
| • polar coordinates | PANG, PRAD |
| • snap to an entity attribute | END, ORG, INT, ON |
| • setting the current location | SET |
| • snap to grid | Grid on (Tools menu) |
| • entity filter | FILTER |

Note: When selecting multiple locations, press [Delete] to take back the locations in the reverse order in which they were selected. In Insert mode, [Delete] also erases the last entity drawn.

Location and Entity Selection

Any time you want to enter exact (absolute) axis coordinates, press [Return] and enter the coordinates in the dialog. Separate each axis value with a comma. For example: 10,100,6. (See Chapter 4: Keyboard Command Entry.)

If you are entering commands by typing, rather than through the icon interface, press [;] after the coordinates are accepted. Press [;] to indicate that the selection process is complete.

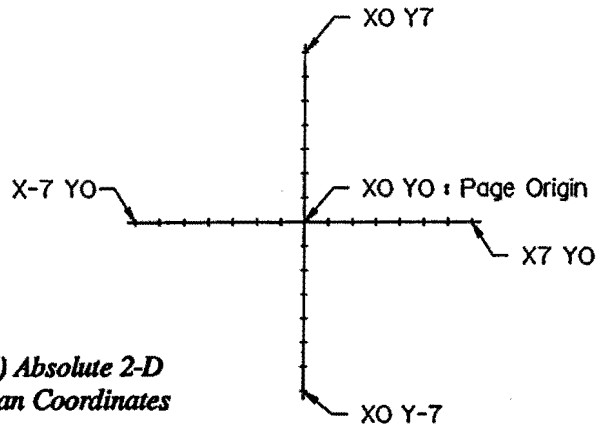


Location by Pointing

Command: LOC [F1]

This is the easiest location selection technique to use. To select a location, place the full screen crosshair at the desired position, and click the mouse button. As with all location selection techniques, a bell sounds if the selected location is invalid. A few points to keep in mind while selecting locations are:

- The location must be within the drawing boundaries.
- The LOC icon must be active to use the X, Y, Z, PRAD, IX, IY, IZ and PANG icons.
- In 3-D mode, the crosshair must be within a defined view.
- Avoid free digitizing in 3-D mode. (see Free Digitizing, located in Chapter 3)
- Click the right mouse button to indicate that your selection is complete.
- Unless tracking is turned off, the current X-Y 2-D coordinates of the crosshair are displayed in the upper right of the screen, above the drawing area.



*fig. (31) Absolute 2-D
Cartesian Coordinates*



Absolute Cartesian Coordinates (2-D)

Command: X, Y

You may reference any 2-D location on the current drawing page by an X and Y location. The X0, Y0 intersection (zero point) is referred to as the page origin and is graphically displayed as a small square with its center at the 0, 0 location:

All locations to the right of this origin are considered positive X locations, and all locations to the left are negative X locations.

All locations above the origin are considered positive Y locations, and all locations below the origin are negative Y locations.

When you request 2-D locations, DynaCADD displays the required icons in the lower menu pad.

To specify an absolute 2-D location:

1. Click the X icon or press [x].
2. Enter the X coordinate into the calculator that pops up.
3. Press [Return] or click the equivalent icon.
4. Click the Y icon or press [y].
5. Enter the Y coordinate into the calculator.
6. Press [Return] or click the equivalent icon.
7. Press [;] or click the equivalent icon.

Absolute Cartesian Coordinates (3-D)

X

Y

Z

Command: X, Y, Z

You can reference any 3-D location in the current design space by an X, Y and Z location. The X0, Y0 and Z0 intersection is referred to as the part origin and is graphically displayed as the center of the Geometric Coordinate Plane (GCP). The three arrows of the GCP always point in the positive X, Y and Z directions.

Note: The actual axis directions are defined by the active GCP. You can change and select the various coordinate planes at any time, even change coordinate systems from within a command.

When you request 3-D locations, DynaCADD displays the required icons in the lower menu pad. To specify an absolute 3-D location:

1. Click the X icon or press [x].
2. Enter the X coordinate into the calculator that pops up.
3. Press [Return] or click the equivalent icon.
4. Click the Y icon or press [y].
5. Enter the Y coordinate into the calculator.
6. Press [Return] or click the equivalent icon.
7. Click the Z icon or press [z].
8. Enter the Z coordinate into the calculator.
9. Press [Return] or click the equivalent icon.
10. Press [;] or click the equivalent icon.

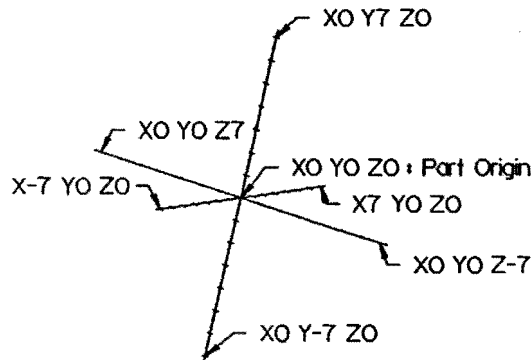


fig. (32) 3-D Coordinate System

Relative Cartesian Coordinates (2-D)**IX****IY****Command: IX, IY**

When you select 2-D locations, the last location of the current command is remembered and referenced as the current location. As an alternative to using strictly absolute coordinates, you can specify relative or incremental coordinates. To specify a relative 2-D location:

1. Click the IX (Increment X) icon or press [i][x].
2. Enter the X distance to move from the current location.
3. Press [Return] or click the equivalent icon.
4. Click the IY (Increment Y) icon or press [i][y].
5. Enter the Y distance to move from the current location.
6. Press [Return] or click the equivalent icon.
7. Press [;] or click the equivalent icon.

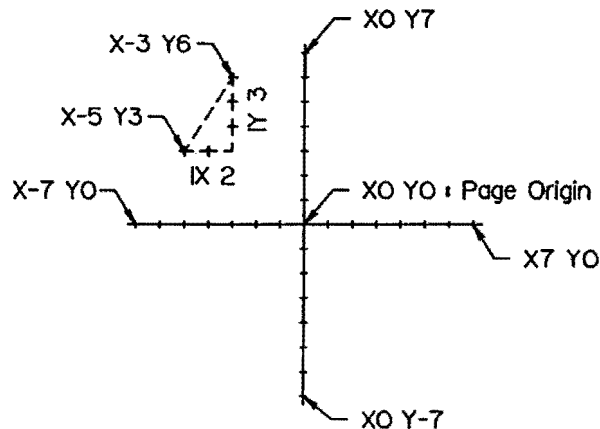


fig. (33) Relative Cartesian Coordinates 2-D

Relative Cartesian Coordinates (3-D)

IX

IY

IZ

Command: IX, IY, IZ

As with 2-D relative locations, the last location of the current command is remembered and referenced to as the current location. To specify a relative 3-D location:

1. Click the IX (Increment X) icon or press [i][x].
2. Enter the X distance to move from the current location.
3. Press [Return] or click the equivalent icon.
4. Click the IY (Increment Y) icon or press [i][y].
5. Enter the Y distance to move from the current location.
6. Press [Return] or click the equivalent icon.
7. Click the IZ (Increment Z) icon or press [i][z].
8. Enter the Z distance to move from the current location.
9. Press [Return] or click the equivalent icon.
10. Press [;] or click the equivalent icon.

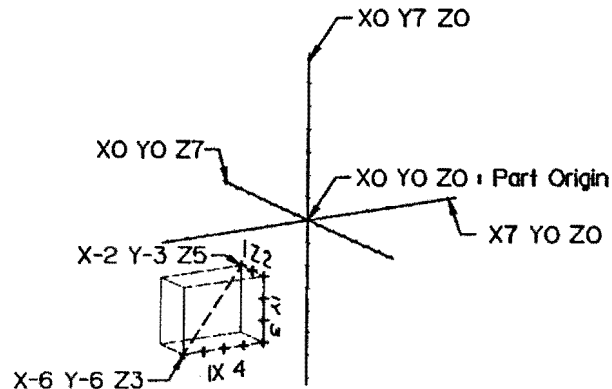


fig. (34) Relative Cartesian Coordinates 3-D

Polar Coordinates (2-D)



Command: PANG, PRAD

Polar coordinates are always considered relative, never absolute. The actual location selected is incremented from the current location.

The polar coordinate system uses both an angle and a radius to determine the location. You must supply the angle in degrees; zero degrees is to the right of the entity origin. The positive angle increases in the counter clockwise direction (0 to 360), negative in the clockwise direction (0 to -360). The radius determines how far from the current location to increment the new location.

To specify a 2-D polar coordinate:

1. Click the Radius icon (PRAD).
2. Enter the distance to move from the current location.
3. Press [Return] or click the equivalent icon.
4. Click the Angle icon (PANG).
5. Enter the angle in degrees (-360 to 360).
6. Press [Return] or click the equivalent icon.
7. Press [:] or click the equivalent icon.

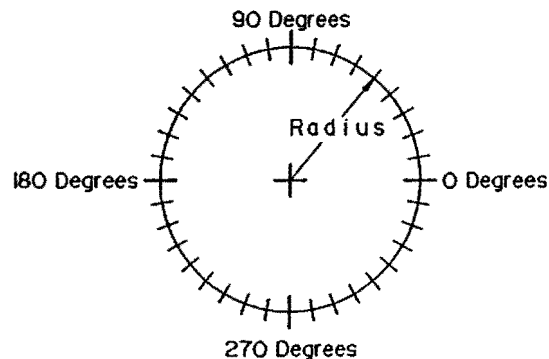


fig. (35) Polar Coordinates (2-D)

Location and Entity Selection

Polar Coordinates (3-D)



Command: PANG, PRAD

3-D polar coordinates operate similar to their 2-D counterparts. The current X-Y plane defined by the active GCP is used to determine the angle. The location is selected using the current Z depth, and rotating about the X-Y plane. As with 2-D polar coordinates, the current location is used as the origin.

To specify a 3-D polar coordinate:

1. Click the Radius icon (PRAD).
2. Enter the distance to move from the current location.
3. Press [Return] or click the equivalent icon.
4. Click the Angle icon (PANG).
5. Enter the angle in degrees (-360 to 360).
6. Press [Return] or click the equivalent icon.
7. Press [;] or click the equivalent icon.

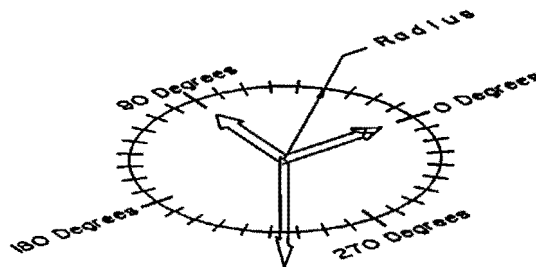


fig. (36) 3-D Polar Coordinates at Z Depth

Location Selection by Entity

Whenever you need to select an entity for a location reference point, the bottom icon pad changes to show the entity selection icons and the drawing crosshair shows two small boxes. These are the entity “traps”; the extent used to determine which entity to select if more than one is near the selection point. The small trap is first checked and, if an entity is found within its boundaries, then it is used. If no entity exists within the small trap, the larger trap is examined. When two or more entities are within a trap, the first entity found is chosen. Use **INFORM LIST LSTDRAW** to show the trap parameters.

Choosing an entity as a selection point is necessary to align entities properly for technical drawings. For example, if you insert a polygon at the origin of a circle, the first vertex of the polygon begins at the 0 degree location of the circle and the origin of the polygon is the same as the origin of the circle.

If entities overlap, click the left mouse button to select the first (topmost), then again to select the next below that, and so on.

With these modifiers, you can select locations via entities by the following methods:

- snap
- end points
- origins
- on or near entities
- intersections
- entity filters

Entity Snap

Entity snap is a very powerful feature that allows locations to reference the attributes of existing geometry. This makes it possible to create very accurate arrangements of entities or parts, without the need to constantly enter the coordinates of each new location. You can reference the following characteristics:

- entity end points
- entity intersections
- entity origins
- on or near entities

With entity snap on, you do not have to worry about free digitizing in 3-D mode (see Chapter 3).

Location and Entity Selection

Entity End Points



Command: END [F2]

Snap a new entity to existing entity end points. All entity types are allowed with this location modifier. To find the end point of any entity type:

1. Click the End icon (END) or press [F2].
2. Select the entity whose end point is required.

Note: When selecting lines, arcs or elliptical arcs, the end closest to the point at which the entity was selected is chosen as the location.

Entity Origins



Command: ORG [F3]

Snap a new entity to existing entity origin or start point. All entity types, except B-spline and Bézier curves, are allowed with this location modifier. To find the origin of any entity type:

1. Click the Origin icon (ORG) or press [F3].
2. Select the entity whose origin is required.

Note: The origin of a line is considered as the midpoint between the two end points.

Entity Intersections



Command: INT [F4]

Snap an entity to the intersection of two existing entities. Lines, arcs and circles are permitted with this modifier. To find the intersection of two entities:

1. Click the Intersection icon (INT) or press [F4].
2. Select the two lines whose intersection point is required.

In 3-D mode, DynaCADD performs a true 3-D intersection. Lines appearing to intersect on the screen may not physically intersect. DynaCADD informs you, should this occur.

Remember when you are selecting entities that the intersection closest to the selection point of the second entity is found. This may appear confusing at first. Examine the example pictures and experiment.

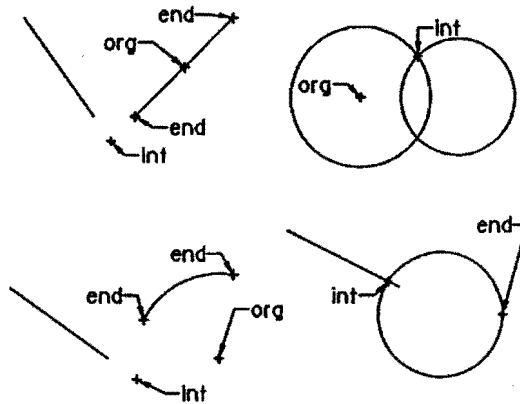


fig. (37) Entity Snap Modes

Entity On



Command: ON

All entity types, except text, B-splines and Bézier curves, are allowed with this location modifier. The location is selected on the chosen entity at the point closest to where the entity was selected. To select a location near an entity:

1. Click the On icon (ON).
2. Select the entity at the point where the location is required.

Setting the Current Location



Command: SET

There are times when you must reference a location without actually selecting that position. For instance, you may need to insert a point three

Location and Entity Selection

units to the right of the origin of a circle. To select a location as the current location:

1. Click the Set icon (SET).
2. Click the Origin icon (ORG) or press [F3].
3. Select the circle by clicking the cross hairs at ent #1.
4. Click the location icon (LOC) or press [F1].
5. Click the increment X icon (IX).
6. Enter the value of 3.0.
7. Press the comma [,] or press the right mouse button.

The selected location is marked with a small cross.

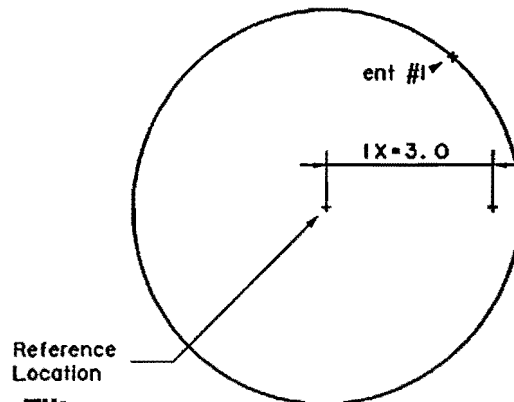


fig. (38)
Set Location

Entity Filters



Command: FILTER

Only certain entities are valid for location and selection commands. All selection techniques allow the selection of "valid" entities. An entity must meet the following conditions to be valid:

- If in 2-D mode, the entity is a 2-D entity.
- If in 3-D mode, the entity is a 3-D entity.
- The layer on which the entity exists is turned "on" and is not locked.
- The appropriate entity filter is "on" (see Entity Filters, below).

You can further enhance all selection techniques by using the entity filter modifier to limit actions to one or more entities based on their type. For example, you can select all line type entities on layer five. To activate entity filters:

1. Click the Entity Filter icon (FILTER).
2. Deselect all entity types on the filter dialog, that you do not want available for selection.
3. Click the OK button.

The Reset button restores the status of all valid entity type buttons to their default condition. The Off button deselects all entity type buttons.

Another use of the entity filter is for information. Since not all commands work on all entities (for example, you cannot fillet a solid), when you select the entity filter icon from the bottom icon pad, the dialog automatically shows which entities are valid for that command. If an entity type is shown as disabled, that specific entity type is not valid in the current command.

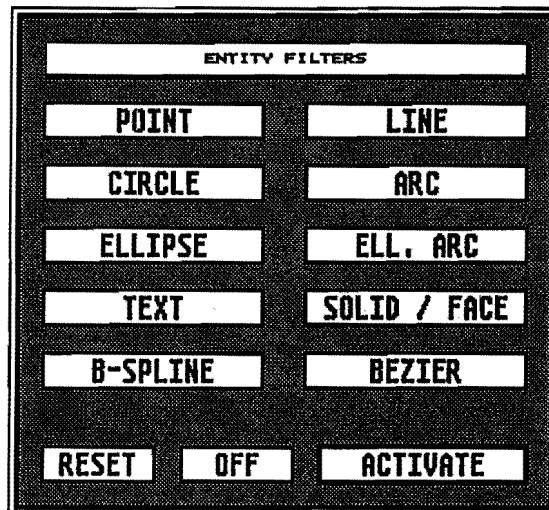


fig. (39) Entity Filter Dialog

Location and Entity Selection

Grid Snap

You can snap all locations, both 2-D and 3-D, to a definable grid. A grid is very similar to graph paper and only allows you to select locations at one of the grid points (intersections). If a location is selected that is not on one of the grid points, the location is snapped to the nearest point. The grid is always considered as 2-D, with the initial grid point (X0, Y0) at the current page origin.

To turn on the grid display and snap, select grid from the Tools menu. Grid is explained in Chapter 5.

Entity Selection

Whenever one or more entities are required, DynaCADD displays the entity selection icons in the bottom icon pad. Many commands permit an unlimited number of entities for transformation, i.e; deleted, copied, mirrored, etc. It is extremely tedious to select the entities you wish the command to act on, not to mention prone to errors, if you were required to manually point and click on every entity. DynaCADD provides the following methods for entity selection:

- | | |
|------------------------|----------|
| • pointing | BYDIG |
| • select all | BYALL |
| • window | BYWIN |
| • window out | BYWOUT |
| • view window | BYVWIN |
| • polywindow | BYPWIN |
| • repetitive last | BYLAST |
| • by layer | BYLAYER |
| • by pen color | BYCOLOR |
| • by pen (line) style | BYSTYLE |
| • by line weight | BYWEIGHT |
| • directional chaining | BYCHAIN |
| • by entity type | FILTER |

As with all entity selection techniques, the valid entities become “dotted”, to indicate which have been selected.

The crosshair changes to show the two entity trap boxes, as described above under Location Selection by Entity.

Note: Once selected, an entity may be deselected by clicking the left mouse button while the entity selection cursor is positioned on the entity and the [Shift] key is held down. This applies to all the following entity selection commands.



Selecting Entities by Pointing

Command: BYDIG [F5]

This is the most common entity selection technique. Pointing is also called "digitizing".

To select the entity, point the full screen crosshair at any position near the desired entity and click the mouse button. If a bell sounds and no entity is selected, DynaCADD was unable to find a valid entity close enough to the selection point. Try again, closer to the desired entity.

Note: Whenever a specific number of entities is required, this is the only valid method of entity selection.



Selecting All Entities

Command: BYALL

There are times when it is easier to simply select all valid entities. Click the BYALL icon.



Selecting Entities Inside a Window

Command: BYWIN [F6]

Click the Window modifier to select entities based on whether they are completely contained within a definable window. To select entities using the window modifier:

1. Click the Window (BYWIN) icon or press [F6].
2. Define the entity window (see Windows, below).

To cancel the window modifier, click the right mouse button or press [;].

Sometimes you are prompted by DynaCADD for a window. A window is a rectangular frame defined by two points diagonally opposite one another. This defines an area within which certain commands are executed or are limited. After you enter the first point, a box follows the cursor, making the actual window visible at all times. Notice the

Location and Entity Selection

difference between a window and a polywindow.

A few important points to remember about windows are:

- All windows are 2-D locations. That is, they exist on the same plane as the screen.
- No part of a window may be off the drawing.
- When you select the two locations, you can use entity snap.



Entities Outside a Window

Command: BYWOUT

Click the Window Out modifier to select entities based on whether they are completely outside a definable window. To select entities using the Window Out modifier:

1. Click the Window Out icon (BYWOUT).
2. Define the entity window (see Windows, below).

Note: To cancel the window out modifier, click the right mouse button or press [;].



Selecting Entities Using View Windows

Command: BYVWIN

Click the View Window modifier to select entities based on whether they are entirely enclosed within a current view definition. This modifier works very similar to the window modifier, with the exception of how the window is defined. To select entities using the View Window modifier:

1. Click the View Window icon (BYVWIN).
2. Select a view (see View Selection, below).

Note: To cancel the view window modifier, click the right mouse button or press [;].

A view is simply a section of a 3-D drawing where you can see parts and entities from a defined angle or perspective. A view clip restricts activity within a selected view (otherwise the drawing page restricts the activity). However, even though on a portion of an entity may be visible in a view, another portion may exist outside the view boundaries. You can change the scale and angle of each view.

If two or more views overlap, DynaCADD chooses the view covering the smallest page area. For example, if you select a segment of overlapping views to delete, then the smallest view is deleted. This insures that all views are selectable at all times.

Up to four views are available on the screen at any time, each with its own angle, zoom, and direction of view. Remember that each view still looks in upon the same drawing and that whatever changes occur in one view occur simultaneously in the others.

Selecting Entities by Polywindows



Command: BYPWIN [F7]

Polywindows are different from regular rectangular windows (see above) in that they permit a definition of an irregular, non rectangular space. To select entities using the Polywindows modifier, click the BYPWIN icon or press [F7].

You create a polywindow by positioning the pointer and clicking the button at any point you wish. A “rubber band” line follows the pointer to indicate the outline of the polywindow (see Rubber Band, Tools Menu, Chapter 5). This line also connects the pointer with the first selected location. The outline can overlap and cross itself. Click the right mouse button when you have all points defined. This “sets” the polywindow.

All entities contained within the polywindow are selected.

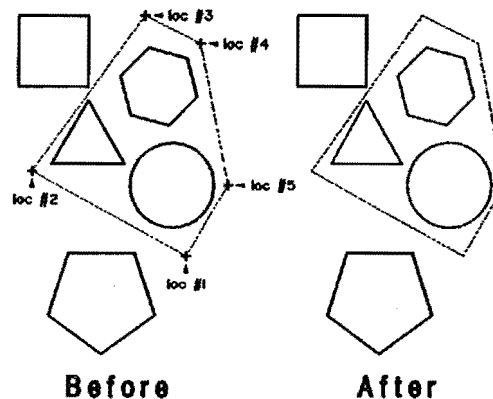


fig. (40) Polywindow

Location and Entity Selection

Selecting the Last Entity



Command: BYLAST

Click the Last modifier to select the last valid entity currently visible on the screen. This is a repetitive modifier, allowing you to select successive entities, based on their position in the database. Click the BYLAST icon to select the last entity.

Selecting Entities by Layer



Command: BYLAYER

Click the Layer modifier to select all entities existing on a specified layer. To select entities by layer:

1. Click the By Layer icon (BYLAYER).
2. Input the required layer number into the pop up calculator.

All entities that are drawn in the requested layer will be selected.

Selecting Entities by Pen Color



Command: BYCOLOR

Click this modifier to select all entities drawn in a specified pen number (which in turn relates to a specific pen color on a plotter). To select entities by pen color:

1. Click the By Pen Color icon (BYCOLOR).
2. Click the required pen number from the dialog.

All entities that are drawn in the requested color will be selected.

Selecting Entities by Style



Command: BYSTYLE

Click this modifier to select all entities drawn in a specified line style. To select entities by style:

1. Click the By Style icon (BYSTYLE).
2. Click the required style named in the dialog window. Use the right and left arrow buttons to scroll the display until the style name appears in the window.

All entities that are drawn in the requested line style will be selected.

Selecting Entities by Weight



Command: BYWEIGHT

Click the By Weight icon (BYWEIGHT) to select entities by their pen weight. Select the weight from the dialog, from 1 (lightest) to 3 (heaviest).

All entities that are visible and have the requested pen weight will be selected.

Selecting Entities by Chaining



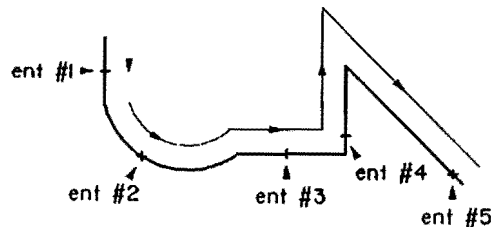
Command: BYCHAIN

Click entity chaining to select entities, based on their connections with other entities. To select entities by chaining:

1. Click the By Chain icon (BYCHAIN).
2. Select the initial entity at the end where the chaining should start.

All entities that share a common end point are selected. If more than two entities share an end point, the direction taken by the chain is undefined.

Note: To cancel the chain modifier, click the right mouse button or press [;].



Entities are selected in successive order. Selecting the first line (ent #1) with the BYCHAIN option causes the arc (ent #2) to be selected, then the horizontal line (ent #3). Entity selection continues in this manner until all the entities chained to the first line (ent #1) are selected.

fig. (41) Selecting Entities by Chaining

Location and Entity Selection

Entity Filters

Command: FILTER

This is the same as described in location selection, above.

Data and Text Entry

Often, a command requires the input of other information in order to execute, including:

- a value for rotation, angle, copies
- a range for layers
- a name for layers
- text for text, dimensions or leaders
- a filename to save or load

When this sort of entry is required, the appropriate dialog, calculator or editor is displayed on the screen.

Inserting Entities ... Chapter 7

The following chapters document the commands and features in DynaCADD in the order in which they appear on the icon pads.

This chapter details the various entity types and insertion techniques available. All Insert Entity commands are combined under the INSERT icon.

Drawings are created primarily by inserting these entities:

- point
- line
- circle
- arc
- fillet (arc)
- ellipse
- elliptical arc
- text
- b-spline curve
- Bézier curve
- 2-D filled (solid) polygon
- hatch symbols
- section lines
- 3-D face

The combination of entities gives you parts and drawings (see Chapter 3). Some of the commands described in this chapter are affected by the settings in the Tools menu (see Chapter 5).

In DynaCADD you create various entities by using the large array of commands and modifiers. These modifiers give you increased flexibility and control over how and where the new entities are created and positioned.

Entities can be modified by using the appropriate transform commands. However, entities on locked or invisible layers cannot be modified until the layer status is changed (see Chapter 15).

The bottom icon pad contains all the in-stream modifiers by which you select entities and locations for the various commands and enter coordinate information (see Chapter 6).

Inserting Entities

Inserting Points

**Primary****Command:** INSERT POINT**Modifiers:** None**Command:** INSERT POINT loc #1**Mode:** 2-D and 3-D**loc #1:** Selected location defining the point origin.**Description:** Points are created and displayed at all selected locations.

A point is a discrete (specific, single) location on the drawing page, represented by a small cross hair, centered on the location. Points are typically used as references for important locations on a drawing, such as the origin of a circle or arc.

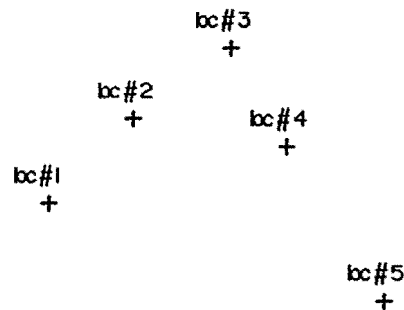


fig. (42) Points

Inserting Lines



Primary

Command: INSERT LINE

Modifiers:

LINELOC



ORTHO



VERTICAL



HORIZONTAL



PERPEND



PARALLEL



TANTO



TANPOINT



LINANG (flag)



LINLEN (flag)



Flags:

These two flags override other line insertion considerations. For example, if you insert a vertical line, but enter a 45 degree angle, the line appears at 45 degrees, regardless of the vertical line icon.

LINANG The angle of the line (–360 to +360 degrees).

LINLEN The absolute length of the line, in the current units.

Inserting Entities

Description: A line is defined by two locations, which you can select and revise by using any one of the numerous modifiers available. Lines are used to define, dimension or as references to various objects on the drawing.

All INSERT LINE commands generate continuous lines, using the end point of the last line as the start point of the next line. A “rubber band” line connects the previous point with the pointer as you move the mouse (see Rubber Band, Tools Menu, Chapter 5). Click the left mouse button to insert a line end point.

The semicolon (;) is used in all INSERT LINE commands to signify that you wish to start a new line without exiting the command. You can also click the right mouse button instead of pressing [;].

Inserting Lines by Location



Command: INSERT LINE LINELOC (flags) loc #1, loc #2

Mode: 2-D and 3-D

Flags: See primary INSERT LINE command.

loc #1: The first (start) point of a line.

loc #2: The second (end) point. Two locations are required to define a line.

Description: A line is created and displayed between locations one and two. You can create a continuous series of lines by selecting additional locations.

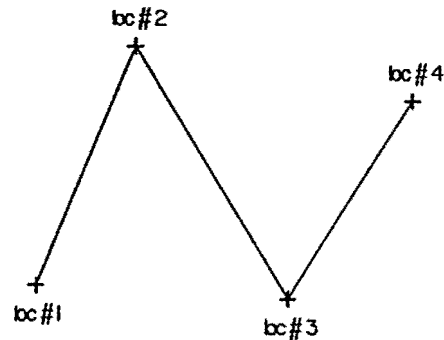


fig. (43) Inserting Lines by Location

Inserting Entities

Inserting Orthogonal Lines



Command: INSERT LINE ORTHO (flags) loc #1, loc #2



Mode: 2-D and 3-D



Flags: See primary INSERT LINE command.

loc #1: The first (start) point of a line.

loc #2: The second (end) point. Two locations are required to define a line.

Description: A vertical or horizontal line is created between the selected locations. The deviation along either the X or Y axis is reduced to zero, depending on the individual deviations between the selected locations. If the distance along the X axis between loc #1 and loc #2 is less than that of the Y distance, the deviation along the X axis is reduced to zero. In 3-D mode, the line is still permitted to travel along the Z axis.

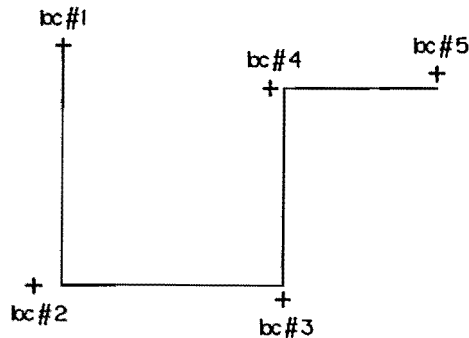


fig. (44) Line, Ortho

Inserting Vertical Lines



Command: INSERT LINE VERTICAL (flags) loc #1, loc #2

Mode: 2-D and 3-D

Flags: See primary INSERT LINE command.

Loc #1: The first (start) point of a line.

loc #2: The second (end) point. Two locations are required to define a line.

Description: A vertical line is created and displayed between the selected locations. The line starts at location #1 and ends at location #2, with any deviation along the X axis reduced to zero. In 3-D mode, a vertical line is still permitted to travel along the Z axis.



fig. (45) Line, Vertical

Inserting Entities

Inserting Horizontal Lines



Command: INSERT LINE HORIZONT (flags) loc #1, loc #2

Mode: 2-D and 3-D

Flags: See primary INSERT LINE command.

loc #1: The first (start) point of a line.

loc #2: The second (end) point. Two locations are required to define a line.

Description: A horizontal line is created and displayed between the selected locations. The line starts at location #1 and ends at location #2 with any deviation along the Y axis reduced to zero. In 3-D mode, a horizontal line is still permitted to travel along the Z axis.

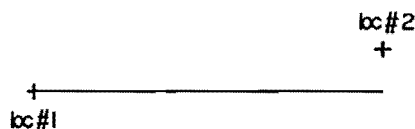


fig. (46) Line, Horizontal

Inserting Perpendicular Lines



Command: INSERT LINE PERPEND (flags) ent #1, loc #1, loc #2

Mode: 2-D and 3-D

Flags: See primary INSERT LINE command.

ent #1: The existing line, used as a reference to define the angle of the new line.

loc #1: The start point of the new line.

loc #2: The end point. Two locations are required to define the line.

Description: A line is created perpendicular to the selected line, starting at location #1 and ending at location #2. If location #2 does not fall exactly on the line, the end of the new line is trimmed normal (perpendicular) to the second location.

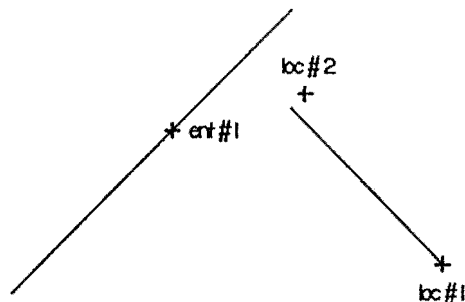


fig. (47) Line, Perpendicular

Inserting Entities

Inserting Parallel Lines



Command: INSERT LINE PARALLEL (flags) ent #1, loc #1, loc #2



Mode: 2-D and 3-D



Flags: See primary INSERT LINE command.

ent #1: The existing line, used as a reference to define the angle of the new line.

loc #1: The start point of a line.

loc #2: The end point. Two locations are required to define the line.

Description: A line is created parallel to the selected line, starting at location #1 and ending at location #2. If location #2 does not fall exactly on the line, the end of the new line is trimmed normal (perpendicular) to the second location.

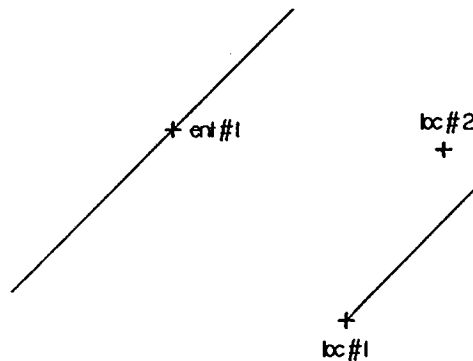


fig. (48) Line, Parallel

Inserting Tangent Lines Between Arcs



Command: INSERT LINE TANTO (flags) ent #1, ent #2

Mode: 2-D only

Flags: See primary INSERT LINE command.

ent #1, ent #2: Any two 2-D arcs, circles or fillets.

Description: A line is inserted tangent to the two selected entities. The selection point for each of the entities determines to which side of the arc the line is drawn.

Note: A continuous series of lines are not available when you use the TANTO modifier.

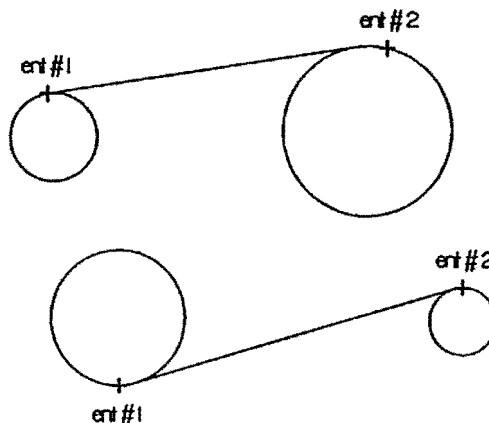


fig. (49) Line, Tangent

Inserting Entities

Inserting Lines Tangent from Locations



Command: INSERT LINE TANPOINT (flags) loc #1, ent #1

Mode: 2-D and 3-D

Flags: See primary INSERT LINE command.

loc #1: The start point of the line.

ent #1: Any arc, circle or fillet.

Description: A line is inserted starting at location #1, tangent to the selected entity. The selection point on the entity determines to which side of the arc the line is drawn.

Note: A continuous series of lines are not available when you use the TANPOINT modifier.

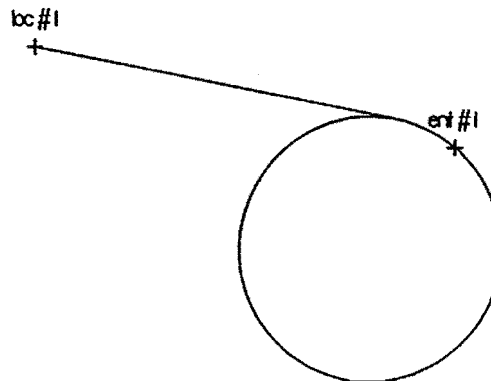


fig. (50) Line, Tanpoint

Inserting Circles



Primary

Command: INSERT CIRCLE

Modifiers:

C2POINT



CIRRAD



CIRDIA



CIRCIRCUM



CBETWEEN



CIRRADLOC



Description:

A circle consists of an origin (center) and a radius. In 3-D mode, all circles are inserted parallel to the current plane defined by the active GCP.

Inserting Entities

Inserting Circles By Origin and Radius



Command: INSERT CIRCLE C2POINT loc #1, loc #2



Mode: 2-D and 3-D



loc #1: The origin, or center, of the circle. This location also defines the Z depth of the circle in 3-D mode.

loc #2: The distance between location #1 and location #2 defines the radius of the circle. This location may be any point on the circumference.

Description: A circle is drawn with its origin at location #1 and a radius large enough so that the circumference passes through location #2.

Note: In 3-D mode, any deviation along the Z axis between the selected locations is ignored.

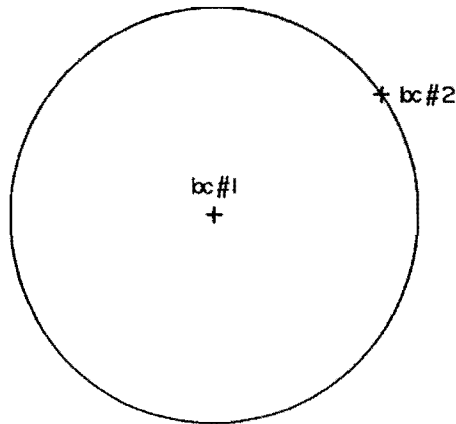


fig. (51) Circle, 2 Point

Inserting Circles by Radius



Command: INSERT CIRCLE CIRRAD (rad) loc #1

Mode: 2-D and 3-D

rad: Numeric value for the radius of the circle.

loc #1: The origin of the circle.

Description: A circle is drawn with its origin at location #1, and a radius of "rad".

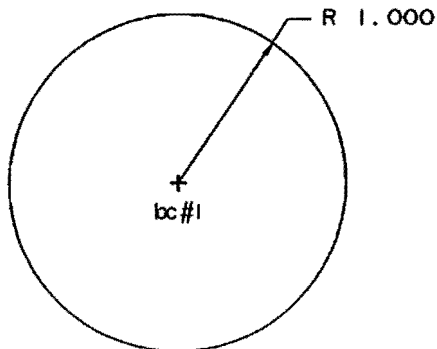


fig. (52) Circle, Radius

Inserting Entities

Inserting Circles by Diameter



Command: INSERT CIRCLE CIRDIA (dia) loc #1



Mode: 2-D and 3-D



dia: Numeric value for the diameter of the circle.

loc #1: The origin of the circle.

Description: A circle is drawn with its origin at location #1, and a diameter of "dia".

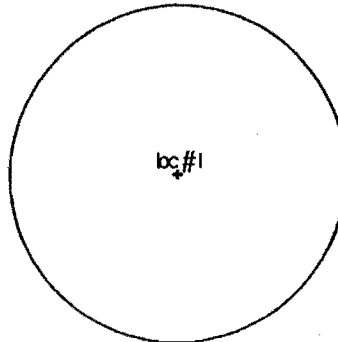


fig. (53) Circle, Diameter

Inserting Circles Using Three Points



Command: INSERT CIRCLE CIRCIRCUM loc #1, loc #2, loc #3



Mode: 2-D and 3-D



**loc #1, loc #2,
loc #3:**

Any three locations on the circumference of the circle.

Description:

A circle is drawn with the circumference of the circle passing through all three locations.

Note:

In 3-D mode, the Z depth of the circle is defined by using the Z depth of location #1. Any deviation along the Z axis between the three locations is ignored.

Slight errors may occur if all three locations are positioned close to each other on the circumference. For best results, space the locations as far apart on the circle as possible.

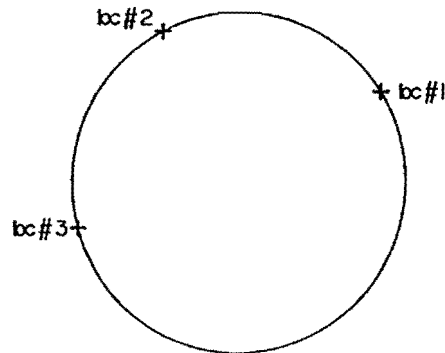


fig. (54) Circle, Circumference

Inserting Entities

Inserting Circles Between Two Points



Command: INSERT CIRCLE CBETWEEN loc #1, loc #2



Mode: 2-D and 3-D



loc #1, loc #2: Any two locations, placed on the circumference of the circle on opposing sides of the origin.

Description: A circle is drawn with its origin at the midpoint between location #1 and location #2. The radius is one half the distance between the two locations.

Note: In 3-D mode, the Z depth of the circle is defined by using the Z depth of location #1. Any deviation along the Z axis between the two locations is ignored.

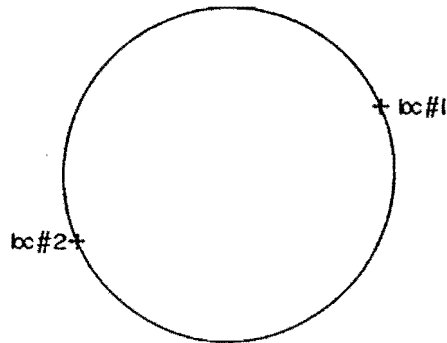


fig. (55) Circle, Between

Inserting Circles of a Specified Radius Between Two Points



Command: INSERT CIRCLE CIRRADLOC (rad) loc #1, loc #2



Mode: 2-D and 3-D



rad: Numeric value for the radius of the circle.

loc #1, loc #2: Any two locations placed on the circumference of the circle.

Description: A circle of radius "rad" is drawn between the two locations, loc #1 and loc #2. If the distance between the two locations is greater than twice the radius "rad", then no circle is drawn.

Note: In 3-D mode, the Z depth of the circle is defined by using the Z depth of location #1. Any deviation along the Z axis between the two locations is ignored.

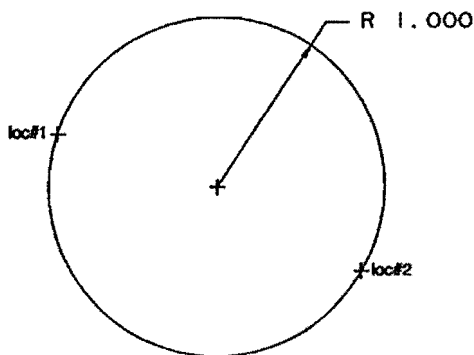


fig. (56) Circle, Radius, Between

Inserting Entities

Inserting Arcs

**Primary****Command:**

INSERT ARC

Modifiers:

ARC3POINT



ARCRAD



ARCDIA



ARCCIRCUM



ARCSTARTA (flag)



ARCENDA (flag)



ABETWEEN



ARCRADLOC

**Flags:**

Where noted, the two angle flags can be used instead of selecting an arc start and end location with the mouse. In both cases, you enter the angle (–360 to 360 degrees) in the pop up calculator. Then you draw the arc as described. The flags are alternatives to locations as described. You must still click at “dummy” locations to fulfill the command, but the values entered into the calculator override any locations selected by clicking the mouse.

ARCSTARTA The arc start angle.

ARCENDA The arc end angle.

Description:

An arc consists of a partial circle defined by an origin, radius, start angle and end angle. The first location is the origin, followed by start and end points.

Note:

All arcs are drawn in a counter clockwise direction, beginning at the start angle and terminating at the end angle. In 3-D mode, all arcs inserted parallel to the current plane defined by the active GCP.

Inserting Arcs Using Three Points



- Command:** INSERT ARC (flags) ARC3POINT loc #1, loc #2, loc #3
- Mode:** 2-D and 3-D
- Flags:** See primary INSERT ARC command.
- loc #1:** The origin of the arc.
- loc #2:** Location #2 defines both the radius and the start angle of the arc.
- loc #3:** Location #3 defines the end angle of the arc.
- Description:** An arc is drawn with its origin at location #1. The arc begins at location #2 and is drawn counter clockwise until the end point of the arc is at the same angle to the arc origin as location #3.
- Note:** The flag setting option of DynaCADD allows increased flexibility in defining arcs. Any flags that are set override the values defined by the selected locations.

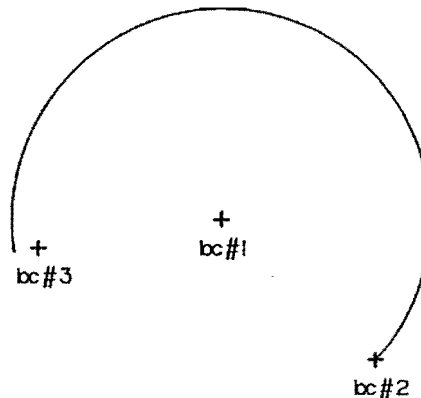


fig. (57) Arc, 3 Point

Inserting Entities

Inserting Arcs by Radius



Command: INSERT ARC (flags) ARCRAD loc #1, loc#2, loc #3



Mode: 2-D and 3-D



Flags: See primary INSERT ARC command.

loc #1: The origin (center) of the arc.

loc #2, loc #3: The start and end points of the arc.

Description: An arc is drawn with a radius equal to the specified value, no matter where you select for the locations.

Enter the radius value into the pop up calculator after you select ARCRAD, then select the three locations that define the arc.

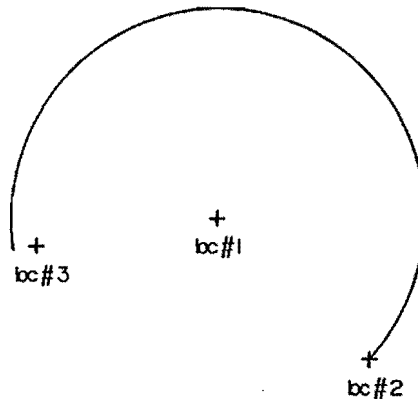


fig. (58) Arc, Radius

Inserting Arcs by Diameter



Command: INSERT ARC (flags) ARCDIA loc #1, loc#2, loc #3

Mode: 2-D and 3-D

Flags: See primary INSERT ARC command.

loc #1: The origin (center) of the arc.

loc #2, loc #3: The start and end points of the arc.

Description: An arc is drawn with the diameter is equal to the specified value, no matter where you select for the locations.

Enter the diameter value into the pop up calculator after you select ARCRAD. Then select the three locations that define the arc.

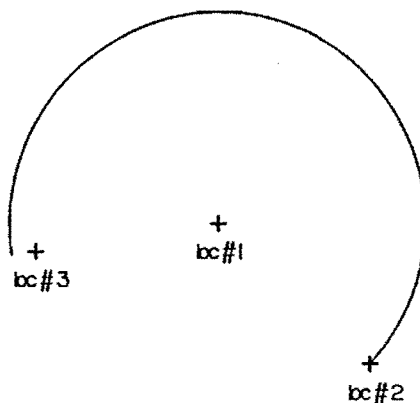


fig. (59) Arc, Diameter

Inserting Entities

Inserting Arcs Using Three Points on Circumference



Command: INSERT ARC (flags) ARCCIRCUM loc #1, loc #2, loc #3

Mode: 2-D and 3-D

Flags: See primary INSERT ARC command.

loc #1, loc #2, loc #3:

Three locations on the circumference of the required arc. Location #1 also defines the start angle, and location #3 defines the end angle.

Description: An arc is drawn with its circumference passing through all three locations. The arc starts at the angle defined by location #1 and is drawn counter clockwise to the angle defined by location #3.

Note: In 3-D mode, the Z depth of the arc is defined by using the Z depth of location #1. Any deviation along the Z axis between the three locations is ignored. Any flags that have been set override the values defined by the selected locations.

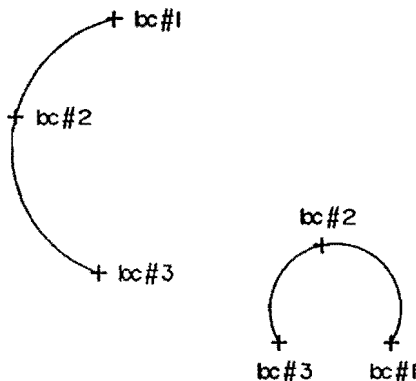


fig. (60) Arc Circumference

Inserting Arcs Between Two Points



Command: INSERT ARC (flags) ABETWEEN loc #1, loc #2

Mode: 2-D and 3-D

Flags: See primary INSERT ARC command.

loc #1, loc #2: Two locations placed on the circumference of the arc. Location #1 is placed at the start of the arc and location #2 is placed at the end point.

Description: An arc is drawn with its origin at the midpoint between the two selected locations. The radius is one half the distance between the two selected points.

Note: The difference between the end angle and the start angle is always 180 degrees when you use this modifier.

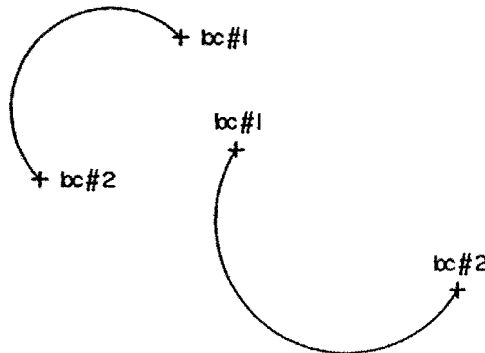


fig. (61) Arc Between Two Points

Inserting Entities

Inserting Arcs of a Specified Radius Between Two Points



Command: INSERT ARC ARCRADLOC (rad) loc #1, loc #2



Mode: 2-D and 3-D



rad: Numeric value for the radius of the arc.

loc #1, loc #2: Any two locations placed on the end points of the arc.

Description: An arc of radius "rad" is drawn between the two locations, loc #1 and loc #2. If the distance between the two locations is greater than twice the radius "rad", then no arc is drawn.

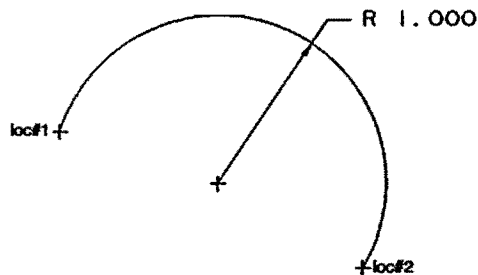


fig. (62) Arc, Radius, Between

Inserting Fillets



Primary

Command: INSERT FILLET

Modifiers:

FILRAD



FILDIA



CFILRAD



CFILDIA



Description:

A fillet is an arc placed between two lines or arcs, with the arc's origin equidistant from those entities. The end points are attached to the start point and end point of the fillet.

INSERT FILLET automatically trims the line ends to meet the ends of the inserted fillet. The selection point on the entity is taken into account when you insert a fillet (see examples, below).

Valid 2-D entities are lines and arcs. In 3-D, the only valid entity is a line.

These commands trim the selected lines or arcs when inserting a fillet. Never select an entity for trimming that is shorter than the radius of the resulting fillet. The results will probably not be what you want.

Note:

Unique to the Insert Fillet command is the ability to bypass the normal insertion of arcs on the plane defined by the current GCP. DynaCADD uses the two selected lines and logically place the fillet's 3-D orientation parallel to the plane defined by these two lines.

A common error message that appears when you use this command in 3-D mode is "No intersection found". The possible causes are:

- no 3-D intersection exists, or
- one of the lines has no deviation along either the current X or Y axis.

Inserting Entities

To solve this problem, select a different GCP and repeat the INSERT FILLET command. In 3-D mode, the entities must also lie on the same plane in a view, parallel to the current GCP, in order for the intersection to work.

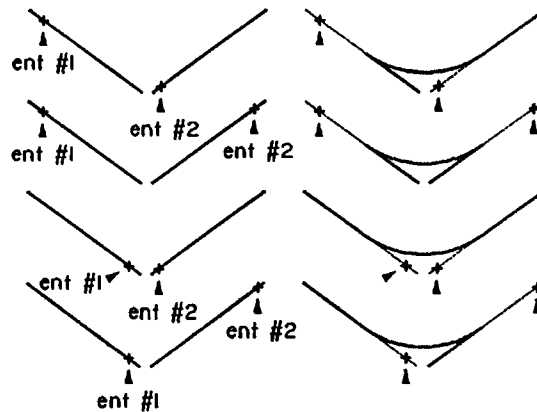


fig. (63) Inserting Fillets by Entity Selection

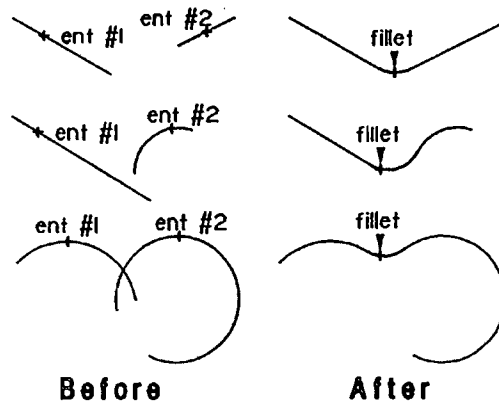


fig. (64) Fillets between Lines and Arcs

Inserting Fillets by Radius



Command: INSERT FILLET FILRAD (rad) ent #1, ent #2

Mode: 2-D and 3-D

rad: The radius of the fillet to insert.

ent #1, ent #2: Two lines used to locate the fillet. These lines must, theoretically, intersect. That is, if they do not intersect, DynaCADD trims them to make them do so.

Description: An arc is drawn with its origin on the intersection of the two lines, an equal distance from the lines themselves. The lines are intersected and physically trimmed a distance of "rad" from the intersection point. If the lines do not physically intersect, DynaCADD projects the position where they intersect (see TRANSFORM TRIM, Chapter 10).

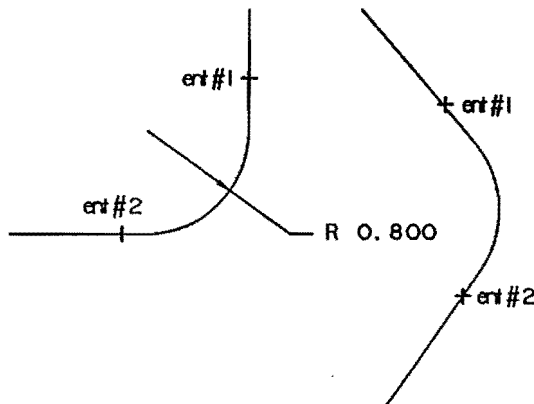


fig. (65) Fillet Radius

Inserting Entities

Inserting Fillets by Diameter



Command: INSERT FILLET FILDIA (dia) ent #1, ent #2

Mode: 2-D and 3-D

dia: Diameter of the fillet to insert.

ent #1, ent #2: Two lines, used to locate the fillet. These lines must, theoretically, intersect (see FILRAD, above).

Description: An arc is drawn with its origin on the bisection of the two lines an equal distance from the lines. The lines are intersected and physically trimmed a distance of one half of "dia" from the intersection point.

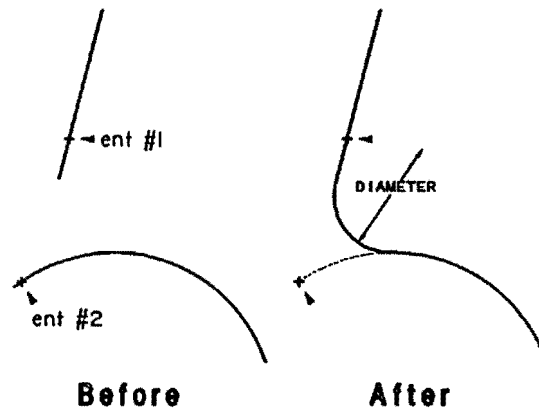


fig. (66) Fillet Diameter

Inserting Corner Fillets by Radius



Command: INSERT FILLET CFILRAD (rad) ent #1, ent #2, ent #3

Mode: 2-D and 3-D

rad: Radius of the fillets to insert.

ent #1, ent #2, ent #3: Three lines on which to perform three successive fillet operations.

Description: INSERT FILLET CFILRAD defines and draws three successive fillets between the three selected lines. The steps DynaCADD follows in order to complete the command are as follows:

1. Fillet and trim lines #1 and #2.
2. Fillet and trim lines #1 and #3.
3. Fillet and trim lines #2 and #3.

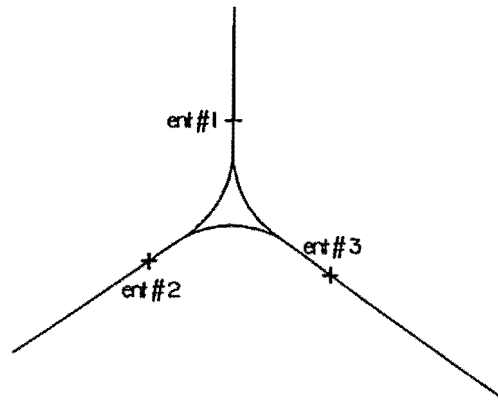


fig. (67) Corner Fillet Radius

Inserting Entities

Inserting Corner Fillets by Diameter



Command: INSERT FILLET CFILDIA (dia) ent #1, ent #2, ent #3

Mode: 2-D and 3-D

dia: Diameter of the fillets to insert.

**ent #1, ent #2,
ent #3:**

Three lines on which to perform three successive fillet operations.

Description: INSERT FILLET CFILDIA defines and draws three successive fillets between the three selected lines. The steps DynaCADD follows in order to complete the command are as follows:

1. Fillet and trim lines #1 and #2.
2. Fillet and trim lines #1 and #3.
3. Fillet and trim lines #2 and #3.

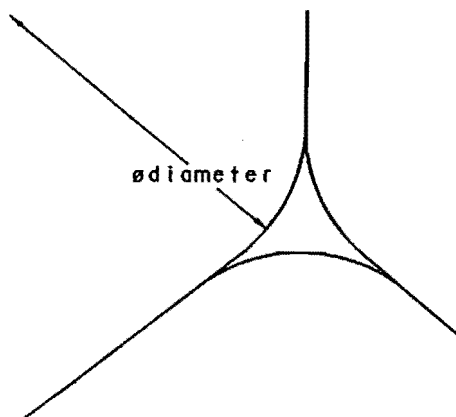


fig. (68) Corner Fillet Diameter

Inserting Ellipses



Primary

Command: INSERT ELLIPSE

Modifiers:

ELLM AJRAD (flag)



ELLM INRAD (flag)



ELLM AJDIA (flag)



ELLM INDIA (flag)



ELLROT (flag)



Flags:

Like the INSERT ARC command, you can enter the values for the flags into the pop up calculator, instead of selecting locations with the mouse. Then draw the ellipse as described. You must still click at “dummy” locations to fulfill the command, but the values entered into the calculator override any locations selected by clicking the mouse.

ELLM AJRAD The major ellipse radius.

ELLM INRAD The minor radius.

ELLM AJDIA The major diameter.

ELLM INDIA The minor diameter.

ELLROT The ellipse rotation.

Description:

The ellipse is very similar to a circle, the difference being that the ellipse has been stretched along one of the axes. The ellipse consists of an origin, a major axis or X radius, a minor axis or Y radius and a rotation about the X Y axes.

In 3-D mode, all ellipses are inserted parallel to the current plane defined by the active GCP.

Inserting Ellipses Using Three Points



Command: INSERT ELLIPSE (flags) loc #1, loc #2, loc #3



Mode: 2-D and 3-D

Flags: See primary INSERT ELLIPSE command.

loc #1: Location #1 defines the origin of the ellipse.



loc #2: Location #2 defines both the semi major axis and the rotation about the X Y axes. As an alternative to loc #2, define the semi major axis through the ELLMAJRAD or ELLMAJDIA flag and the rotation (-360 to +360 degrees) through the ELLROT flag.



loc #3: Location #3 defines the semi minor axis. The semi minor axis is the normal (perpendicular) distance of location #3 to an imaginary line between location #1 and location #2. As an alternative to loc #3, define the minor axis by using either the ELLMINRAD or ELLMINDIA flag.

Description: An ellipse is drawn with its origin at location #1. The semi major axis is the distance between location #1 and location #2. The rotation of the ellipse is equal to the angle between location #1 and location #2. The semi-minor axis is the normal (perpendicular) distance of location #3 to the imaginary line defined by location #1 and location #2.

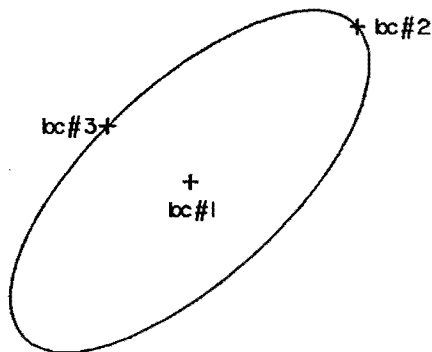


fig. (69) Ellipse

Inserting Elliptical Arcs



Primary

Command: INSERT ELLARC

Modifiers:

ELAMAJRAD (flag)



ELAMINRAD (flag)



ELAMAJDIA (flag)



ELAMINDIA (flag)



ELAROT (flag)



ELASTARTA (flag)



ELAENDA (flag)



Flags:

Like the INSERT ELLIPSE command, you can enter the values for the flags into the pop up calculator, instead of selecting locations with the mouse. Then draw the elliptical arc as described. You must still click at “dummy” locations to fulfill the command, but the values entered into the calculator override any locations selected by clicking the mouse.

ELAMAJRAD The major ellipse radius.

ELAMINRAD The minor radius.

ELAMAJDIA The major diameter.

ELAMINDIA The minor diameter.

ELAROT The ellipse rotation.

ELASTARTA The start angle of the arc.

ELAENDA The end angle of the arc.

Description:

An elliptical arc consists of an origin, a major axis or X radius, a minor axis or Y radius, a start angle and an end angle. The elliptical arc may also have a rotation about the X Y axes applied to it.

In 3-D mode, all elliptical arcs are inserted parallel to the current plane defined by the active GCP.

Inserting Entities

Inserting Elliptical Arcs Using Five Points



Command: INSERT ELLARC (flags) loc #1, loc #2, loc #3, loc #4, loc #5

Mode: 2-D and 3-D

Flags: See primary INSERT ELLARC command.

loc #1: Location #1 defines the origin of the elliptical arc.



loc #2: Location #2 defines both the semi major axis and the rotation about the X Y coordinate axes. An alternative to loc #2, define the semi major axis by using either the ELAMAJRAD or ELAMAJDIA flags. Define the rotation about the X Y coordinates axes (-360 to $+360$ degrees) by using the ELAROT flag.



loc #3: Location #3 defines the semi minor axis. The semi minor axis is the normal (perpendicular) distance of location #3 to an imaginary line between location #1 and location #2. AS an alternative to loc #3, define the minor axis by using either the ELAMINRAD or ELAMINDIA flag.

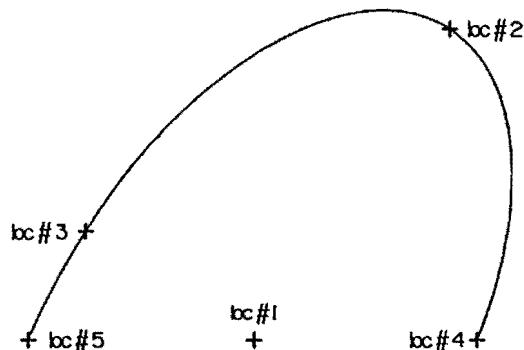


fig. (70) Elliptical Arcs



loc #4:

Location #4 defines the start angle of the elliptical arc.
As an alternative to loc #4, define the start angle by using the ELASTARTA flag.



loc #5:

Location #5 defines the end angle of the elliptical arc.
As an alternative to loc #5, define the end angle by using the ELAENDA flag.

Description:

An elliptical arc is drawn with its origin at location #1. The semi-major axis is the distance between location #1 and location #2. The rotation of the ellipse is equal to the angle between location #1 and location #2. The semi-minor axis is the normal (perpendicular) distance of location #3 to the imaginary line defined by location #1 and location #2. The elliptical arc is drawn in a counter-clockwise direction until the end point is at the same angle to the origin as that of location #5.

Inserting Entities

Inserting Boxes



Primary

Command: INSERT BOX

Modifiers: None

Command: INSERT BOX loc #1, loc #2

Mode: 2-D and 3-D

loc #1, loc #2: Selected locations defining the two diagonally opposing corners of the box.

Description: INSERT BOX creates four individual lines, defined by two diagonally opposite locations. The lines are not grouped, and you can reference them individually, during the design process.

Four lines are inserted in the form of a rectangle, defined by the selected, diagonally opposite locations.

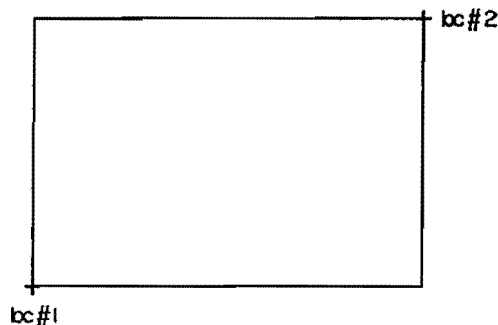


fig. (71) Box

Inserting Polygons



Primary

Command: INSERT POLYGON

Modifiers: POLYSIDES (flag)

POLYROT (flag)

POLYRAD (flag)

POLYDIA (flag)

POLYAFLAT (flag)



Flags: **POLYSIDES** The number of sides to the polygon.

POLYROT The rotation about the X Y axis
(-360 to 360 degrees).

POLYRAD The radius of the polygon.

POLYDIA The diameter of the polygon.

POLYAFLAT The distance between 2 opposing flat
sides of the polygon.

Note: The default number of POLYSIDES is set at six.
Whenever the distance across flats of the polygon is set,
using the POLYAFLAT modifier, the number of sides
should be an even number.

Description: A polygon consists of a definable number of sides,
grouped a common distance from the origin (the lines
form a group). In 3-D mode, all polygons are inserted
parallel to the current plane defined by the active GCP.

Inserting Entities

Inserting Polygons Using Two Points



Command: INSERT POLYGON (flags) loc #1, loc #2



Mode: 2-D and 3-D

Flags: See primary INSERT POLYGON command.

loc #1: Location #1 defines the origin of the polygon.

loc #2: Location #2 defines both the radius and the rotation about the X Y coordinate axes.



Description: A Polygon is drawn with its origin at location #1. The radius is the distance between location #1 and location #2. The rotation of the polygon is equal to the angle between location #1 and location #2. You enter the number of sides of the polygon by using the POLYSIDES modifier, initially set to six.

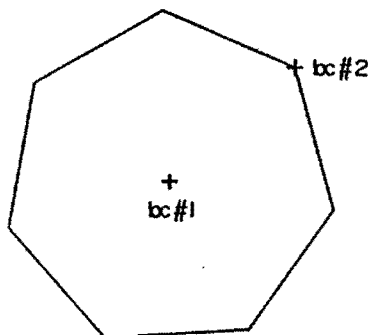


fig. (72) Polygons

Inserting Text



Primary

Command: INSERT TEXT

Modifiers:

TEXTWID (flag)



TEXTHEI (flag)



TEXTROT (flag)



TEXTSLANT (flag)



SETPSPACE (flag)



TEXTLEFT (flag)



TEXTRIGHT (flag)



TEXTCENT (flag)



TEXTLOC



TEXTDIGI



TEXTASSUM



TEXT2ROT



Flags:

You can change the default parameters - width, height, slant and rotation - at any time from within the INSERT TEXT command. Once new text definitions have been selected, the new definitions become the default parameters.

To change the parameters of existing text, including the text itself, select the appropriate commands from the TRANSFORM MODIFY icon pad.

TEXTWID and TEXTHEI settings also affect Dimension text (see Chapter 9).

Note:

When setting the text width and height in the drawing, DynaCADD assumes that the value given is either in inches or millimeters exclusively, depending on the currently selected unit type, English or Metric,

Inserting Entities

respectively. The selected value is automatically readjusted to the drawing scale.

TEXTWID	Defines the width of a single character box.
TEXTHEI	Defines the height of a single character box. This determines the font height and is based on the distance from the baseline to the top of the uppercase "A".
TEXTROT	Defines the rotation of the text about the current X Y coordinate axes (−360 to +360 degrees).
TEXTSLANT	Defines the slant (−60 to +60 degrees) applied to each character in the text. A positive value slants the characters to the left (backward). Normally, slanted text such as italic, slants to the right 10 to 12 degrees, entered as negative value.
SETPSPACE	Defines the character spacing of the text. This is the same as in the TRANSFORM MODIFY command. After you select the SETPSpace icon, a small dialog box appears with the following options:
Constant	Click here for constant, uniform character spacing. Characters are measured according to their full box size, as defined in the Vector Font Editor, taken from the letter "A". Manual spacing affects this value (see below).

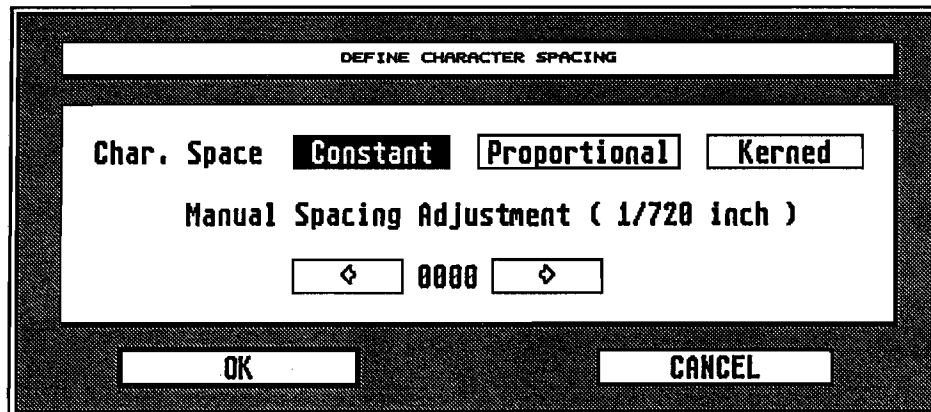


fig. (73) SETSPACE Dialog

Inserting Entities

Note: Text Justification (left, right, center) is retained when you use the TRANSFORM MODIFY TXTORIENT command. See Chapter 10.

When inserting text with multiple lines (separated by pressing [Return] in the text editor), each line is broken into a separate entity.

Default Setup:	WIDTH	0.100 Inches
	HEIGHT	0.125 Inches
	ROTATION	0.0 degrees
	SLANT	0.0 degrees
	LEFT	justification

Description: Text consists of an origin, width, height, character spacing, rotation and slant. In addition to this, each text string has its own type style and font assigned to it. In 3-D mode, all text is inserted on the current plane defined by the active GCP.

The maximum length of the text string is 800 characters. You can create multiple lines by pressing [Return] to end the current line and begin a new line. Press [Enter] on the keypad to finish, [Esc] or [Undo] to cancel.

All text is entered through the text editor. See Chapter 1 for more information.

Note: You must select a font before inserting text. Each new text string is assigned the currently active font style.

Character spacing is normally proportional, but can be changed by selecting the TRANSFORM MODIFY CHATSPACE command.

See Chapter 1 for a description of how to use the text editor.

Inserting Text by Location



Command:	INSERT TEXT (flags) TEXTLOC (string) loc #1
Mode:	2-D and 3-D
string:	This is the actual text to insert, as a string (or strings, if multiple lines are entered).
loc #1:	This location defines where the text is placed. How the text is placed in relation to this location depends on the currently active justification.
Description:	You create and edit the text in the text editor. This text is drawn with the currently active font style.

+ TEST STRING...
bc #1

fig. (75) Text Location

Inserting Entities

Inserting Text Using Three Points (Digitize)



Command:	INSERT TEXT TEXTDIGI (string) loc #1, loc #2, loc #3
Mode:	2-D and 3-D
string:	This is the actual text to insert, as a string (or strings, if multiple lines are entered).
loc #1:	Location #1 defines the lower left corner of the text block.
loc #2:	Location #2 defines both the width of the text block and the rotation of the text block.
loc #3:	Location #3 defines the height of the text block. This height is the normal (perpendicular) distance from location #3 to an imaginary line between location #1 and location #2.
Description:	Text consisting of the characters defined by "string", inserted to fit into an imaginary box defined by the three locations. This method is extremely useful when fitting text blocks into a defined area. The current text slant is applied to the inserted text.

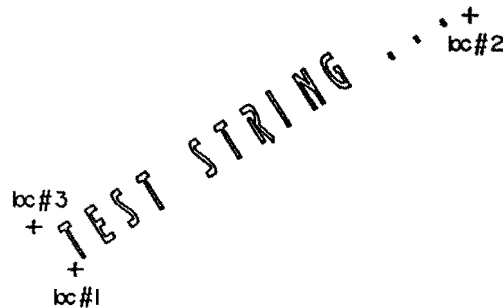
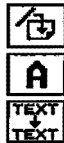


fig. (76) Text Digitize

Assuming Existing Text Attributes



Command:	INSERT TEXT TEXTASSUM (string) ent #1, loc #1
Mode:	2-D and 3-D
string:	This the actual text to insert, as a string (or strings if multiple lines are entered).
ent #1:	Existing text used to define the width, height, rotation and slant of the text to insert.
loc #1:	Location #1 is the lower left corner of the text block to insert.
Description:	The text is drawn with the lower left corner of the text block at location #1. The width, height, rotation and slant assume those of entity #1.
Note:	The currently active justification is used to position the text relative to location #1.

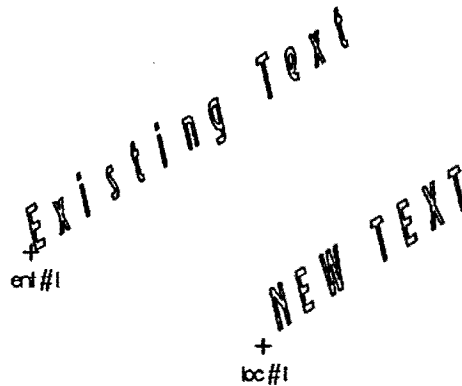


fig. (77) Text Assume

Inserting Entities

Inserting Text in 2-D Rotation



Command: INSERT TEXT TEXT2ROT loc #1 loc #2



Mode: 2-D and 3-D



loc #1, loc #2: The start and end point of the baseline on which the text appears.

Description: The text appears at the angle dictated by the rotation angle between location #1 and location #2. The current justification, character height, width and slant are unchanged.

Text Using 2 Locations

+ loc #1

loc #2

fig. (78) Text in 2-D Rotation

Inserting Subfigures



Primary

Command: INSERT SUBFIG

Modifiers:

SFIGROT (flag)



SFIGXSCA (flag)



SFIGYSCA (flag)



SFIGZSCA (flag)



Flags:

The scale affects the size of the subfigure when inserted. A scale of 0.5 is interpreted as one half the original size, and a scale of 2.0 is interpreted as twice the original size. All scaling factors are applied before any rotation.

A negative value entered for a scaling flag mirrors the subfigures around the appropriate axis, at that scale factor. For example, SFIGXSCA -2 means that the figure will be twice the scale on the X axis, and mirrored about that axis.

The following factors will be applied to the original entity definition :

SFIGROT The rotation (-360 to 360 degrees) about the insertion point.

SFIGXSCA A scaling factor applied to the X axis.

SFIGYSCA A scaling factor applied to the Y axis.

SFIGZSCA A scaling factor applied to the Z axis.

Description:

Subfigures are groups of entities that were previously created and saved in a special figure file (see CREATE DEFFIG, Chapter 12). They can be saved and loaded to disk. Upon insertion, all entities are grouped and acted upon as a single entity.

Note:

You cannot insert figures created from 3-D entities into the drawing as 2-D entities or figures created from 2-D entities into the drawing as 3-D entities.

The Inheritance Option affects this command. (See Chapter 5, page 62.)

Inserting Subfigures by Location



Command: INSERT SUBFIG name (flags) loc #1



Mode: 2-D and 3-D

Name: After you select the SUBFIG icon, a File Selector box appears. The file extensions for valid figure types for insertion are:

.FIG
.PRT
.DPT

loc #1: Location #1 is the insertion point on the current drawing. A matching insertion point on the figure is selected when you create the figure.

Description: A previously created figure (see CREATE DEFFIG, Chapter 12) is inserted with its origin (the insertion point selected when created) at location #1.



The figure's size is scaled along the individual axes using the scaling flags SFIGXSCA, SFIGYSCA and SFIGZSCA. If a negative scaling factor is entered, then the subfigure is flipped on that axis when inserted.

The figure is rotated SFIGROT degrees about the insertion point. All entities are grouped upon insertion and treated as if they were a single complex entity (polyfigures, below, are broken into their component entities).

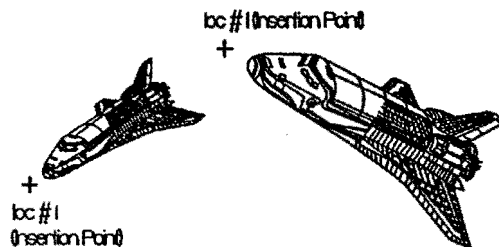


fig. (79) Subfigures

Inserting Polyfigures



Primary

Command: INSERT POLYFIG

Modifiers:

PFIGROT (flag)



PFIGXSCA (flag)



PFIGYSCA (flag)



PFIGZSCA (flag)



Flags:

A scale of 0.5 is interpreted as one half the original size, and a scale of 2.0 is interpreted as twice the original size. All scaling factors are applied before any rotation.

PFIGROT The rotation (–360 to 360 degrees) about the insertion point, applied to the figure.

PFIGXSCA A scaling factor applied to the X axis of the original entity definitions.

PFIGYSCA A scaling factor applied to the Y axis of the original entity definitions.

PFIGZSCA A scaling factor applied to the Z axis of the original entity definitions.

Description:

Polyfigures are groups of entities that were previously created and saved in a special figure file. Upon insertion, all entities have their individual status restored (subfigures, above, remain as one single, complex entity).

Note:

You cannot insert figures created from 3-D entities into the drawing as 2-D entities or figures created from 2-D entities into the drawing as 3-D entities.

The Inheritance Option affects this command. (See Chapter 5, Page 62)

Inserting Entities

Inserting Polyfigures by Location



Command: INSERT POLYFIG name (flags) loc #1

Mode: 2-D and 3-D

Name: After you select the POLYFIG icon, a File Selector box appears. The file extensions for valid figure types for insertion are:

.FIG

.PRT

.DPT

loc #1: Location #1 is the insertion point on the drawing. The matching insertion point on the figure was previously defined when you create the figure.



Description: A previously created figure is inserted with its origin at location #1. The figure's size is scaled along the individual axes, by using the scaling flags PFIGXSCA, PFIGYSCA and PFIGZSCA. The figure is rotated PFIGROT degrees about the insertion point. All entities are released to their individual status.

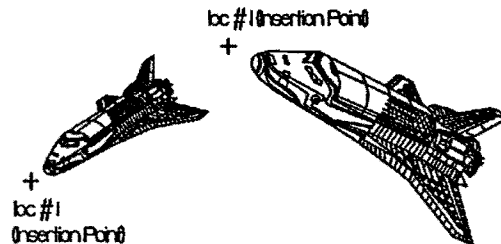


fig. (80) Polyfigures

Inserting Sections



Primary

Command: INSERT SECTION

Modifiers:

SECTDIST (flag)



SECTANGLE (flag)



Flags:

SECTDIST The absolute distance between section lines, in the current units.

SECTANGLE The angle (−360 to +360 degrees) for the section lines.

Description:

Draws section lines at the specified distance and at the specified angle, between all selected entities. All entities must be connected and form an enclosed area. The section lines are treated as one entity. In 3-D mode, section lines are drawn on the current plane.

The section lines are drawn in the current pen, weight and style. Usually section lines are thin and drawn at 45 degrees to the major outline.

The overlap of two entities is an “island” and does not receive sections lines, but the intersection of three does. Even numbered intersections (two entities, four entities, etc.) are not sectioned, but odd numbered are sectioned.

Sections are not unique entities; they are groups of lines. To remove or modify individual section lines, use the CREATE UNGROUP command.

Inserting Entities

Inserting Section Lines



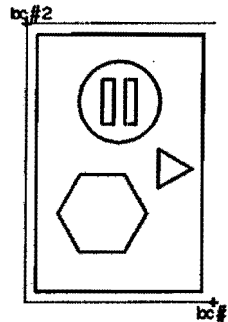
Command: INSERT SECTION (flags) ent #1...ent #n

Mode: 2-D and 3-D

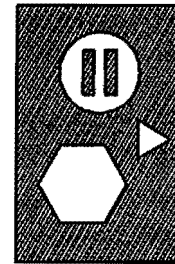
ent #1...ent #n: Entities selected to receive section lines. All entities must be made of contiguous lines that form a completely enclosed area.

Description: Click the right mouse button or press [.] to activate this command. See above.

Press the right mouse button to halt the sectioning process.



Before



After

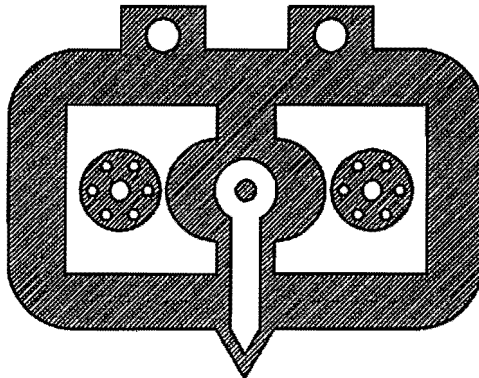


fig. (81) 2 Section Line Examples

Inserting Hatches



Primary

Command: INSERT HATCH

Modifiers: HATCHX (flag)



HATCHY (flag)



Flags:

HATCHX The scale factor for the hatch pattern, in the X axis.

HATCHY The scale factor for the hatch pattern, in the Y axis.

Description:

Draws hatch patterns on the current plane, at the specified X and Y scales, inside all selected entities. All entities must be connected as contiguous lines and form an enclosed area. The hatch marks are treated as one entity.

The hatch marks are drawn in the current pen, weight and style.

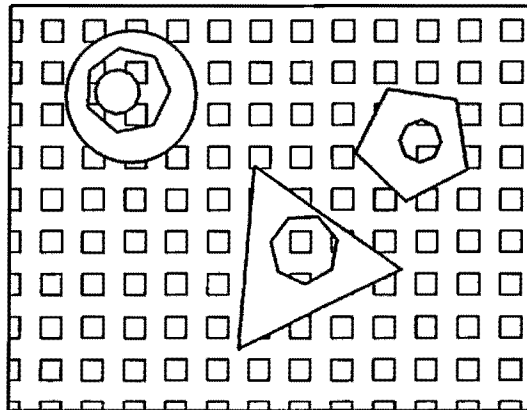


fig. (82) *Overlapping Hatch Marks*

Inserting Entities

The overlap of two entities does not receive hatch lines, but the intersection of three does. Even numbered intersections (two entities, four entities, etc.) are not hatched, but odd numbered are hatched.

Choose the current hatch pattern by selecting Hatch Pattern from the Set menu. See Chapter 5 for more information.

Hatch marks are not unique entities; they are groups of lines. To remove or modify individual lines, use the CREATE UNGROUP command.

See the notes on creating a hatch pattern in Chapter 16.

Inserting Hatch Marks



Command: INSERT HATCH (flags) ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: Entities selected to receive hatch marks. All entities must be made of continuous lines and form a completely enclosed area.

Description: Click the right mouse button or press [;] to activate this command.

Hatch marks are created from the patterns in the HATCH.FNT font file. These patterns can be modified in the Vector Font Editor. Hatch symbols are commonly used to identify terrain or material type.

Press the right mouse button to halt the hatching process.

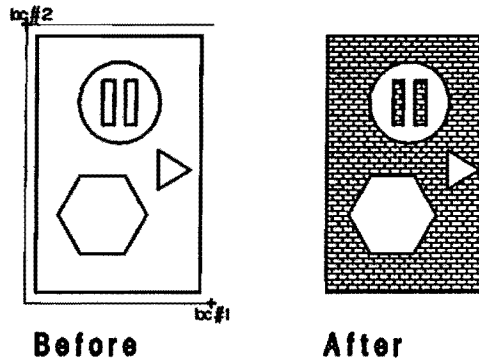


fig. (83) Hatch Marks

Inserting Entities

Inserting Solid 2-D Entities



Primary

Command: INSERT SOLID

Modifiers: None

Command: INSERT SOLID loc #1, loc #2, loc #3, loc #4

Mode: 2-D

loc #1...loc #4: Locations that define the polygon corners. The fourth location is automatically joined to the first and the area filled in.

The fourth location is optional. If you click the right mouse button after the third location is selected, it is joined to the first.

Description: Insert a solid (filled) four sided polygon. This is a continuous process, similar to inserting lines, with subsequent polygons inserted from locations defined for the previous polygon. Filled polygons are often used for solid arrowhead symbols.

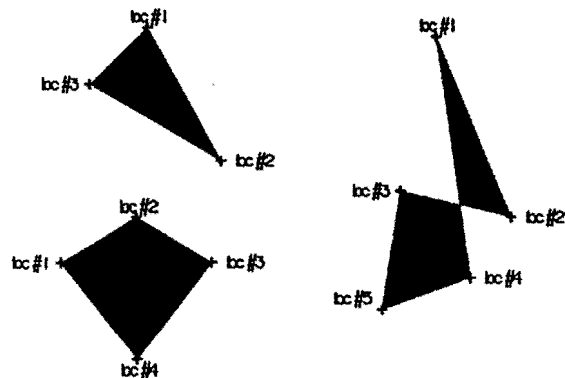


fig. (84) Solids

The 2-D solid is drawn when the fourth location is selected. The two last locations then become the first and second locations, respectively, for another solid polygon which requires only two more locations. The second location of the next polygon is automatically connected to the third location of the previous polygon and filled.

Press [Delete] to delete the last filled polygon(s), one at a time.

Inserting Entities

Inserting 3-D Face Entities

**Primary****Command:** INSERT FACE**Modifiers:** None**Command:** INSERT FACE loc#1, loc#2, loc#3, loc#4;**Mode:** 3-D

loc#1...loc#4: Locations that define the face corners. The fourth location is automatically joined to the first and the area filled in. The fourth location is optional. If you click the right mouse button after the third location is selected, it is joined to the first.

Description: Insert a four sided face. This is a continuous process, similar to inserting lines, with subsequent faces inserted from locations defined for the previous face.

The 3-D face is drawn when the fourth location is selected. The two last locations then become the first and second locations, respectively, for another 3-D face which requires only two more locations. The second location of the next face is automatically connected to the third location of the previous face.

Press [Delete] to delete the last filled face(s), one at a time.

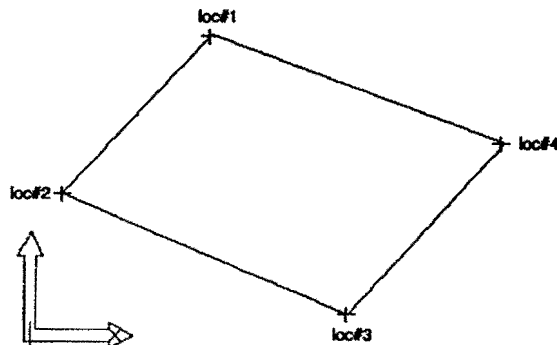


fig. (85) 3-D Face

Deleting Entities



Primary

Command: INSERT DELETE

Modifiers: None

Command: INSERT DELETE ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: Selected entities to delete. All entity types work with this command.

Description: Just as important as inserting entities is the ability to delete them. Click on the Delete icon and use one of the entity selection methods described in Chapter 5, to choose the entity or entities to delete.

You can “Undo” a delete command.

Click the right mouse button or press [.] to activate this command.

Note: You can also press [Delete] to delete the last entity drawn. If the entity is a “chained” entity such as continuous lines or filled solids, [Delete] erases the last entity, one at a time, in reverse order.



Curves ... Chapter 8

B-spline and Bézier curves are special entities in DynaCADD. They are created by first drawing "control points" on the screen, then calculating the curve to fit those points.

B-spline and Bézier curves are built using different algorithms and thus generate different curve plots. Essentially, B-spline curves provide a closer curve fit along the path dictated by the control points, while Bézier curves are a smoother relationship between the individual control and end points.

Two options in the Tools menu affect curves:

Graphics This determines whether the display of the curve construction lines and direction arrows appear on screen. If you are having trouble selecting curves, turn on Graphics and click on the construction lines.

Close Curve If active, this allows the last location selected to be joined to the first when you click the right mouse button.

Curves are created in the flow direction from the first to last control point.

All curve commands are grouped under the CURVE icon.

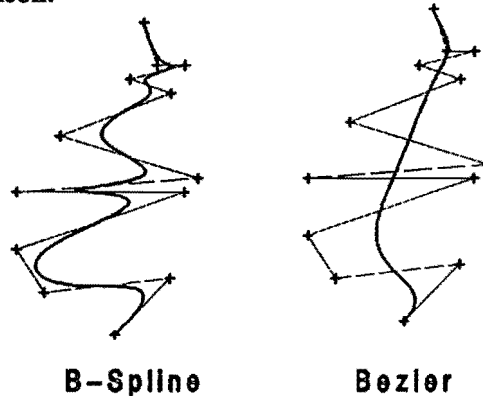


fig. (86) B-spline and Bézier Curves based on Identical Control Points

Inserting B-spline Curves



Primary

Command: CURVE BSPLINE loc #1... loc #n

Mode: 2-D and 3-D

loc #1..loc #n: The location for the curve control points. At least three control points are necessary to make a B-spline curve. The maximum is 49.

Description: Draws a B-spline curve based on the location of the control points. Curve construction lines can overlap each other.

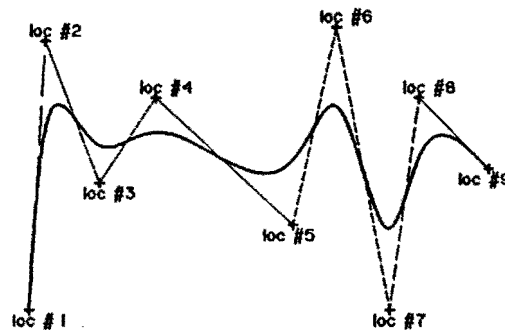


fig. (87) B-spline Curve

Inserting Bézier Curves



Primary

Command: CURVE BEZIER loc #1... loc #n

Mode: 2-D and 3-D

loc #1...loc #n: The location for the curve control points. At least four control points are necessary to make a Bézier curve. The maximum is 12. The more control points inserted, the longer it takes to plot the curve.

Description: Draws a Bézier curve based on the location of the control points. Curve construction lines can overlap each other.

CURVE BEZIER loc #1...loc #9 ;

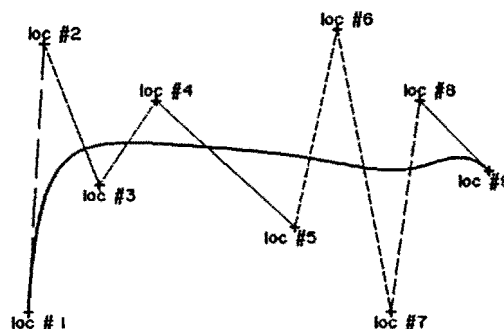


fig. (88) Bézier Curve

Changing Curve Type



Primary Command: CURVE CONVERT

Modifiers: TOBEZIER
TOBSPLINE



Description: Changes the selected curve from one type to the other, depending on the conversion flag.

Changing B-Spline Curves to Bézier Curves



Command: CURVE CONVERT TOBEZIER ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: Curves to convert.

Description: Changes the selected B-spline curve(s) to Bézier curve(s). Click the right mouse button, or press [.] to activate this command.

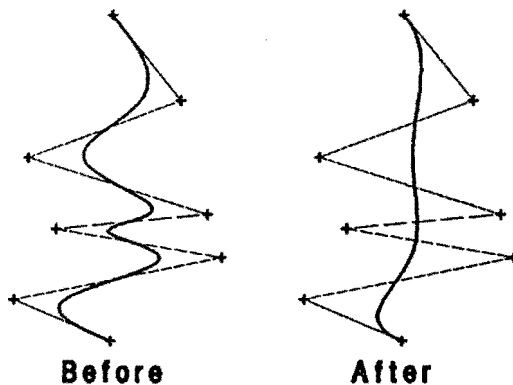


fig. (89) B-spline to Bézier Curve

Changing Bézier Curves to B-Spline Curves



Command: CURVE CONVERT TOBSPLINE ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: Curves to convert.

Description: Changes the selected Bézier curve(s) to B-spline curve(s). Click the right mouse button, or press [:] to activate this command.

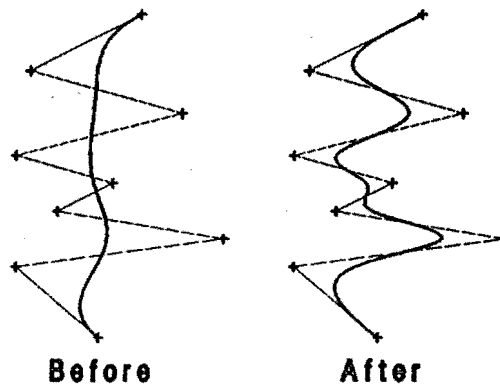


fig. (90) Bézier to B-spline Curve

Breaking Curves



Primary

Command: CURVE BREAK

Modifiers: None

Command: CURVE BREAK ent #1

Mode: 2-D and 3-D

ent #1: The control point on the curve to break. DynaCADD chooses the closest control point, if one is not selected.

Description: Breaks the selected curve at the indicated control point.

The curve is broken into two curves at the selected control point. The control point is duplicated, one for each curve. Both new curves share the same direction of flow as the parent curve. The two curves must both have the requisite number of control points when broken, else the break does not occur.

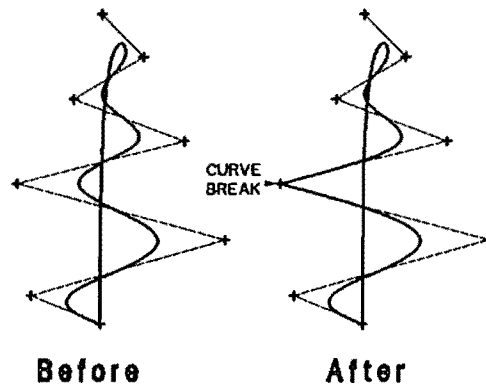


fig. (91) Curve Break

Joining Two Curves



Primary

Command: CURVE JOIN

Modifiers: None

Command: CURVE JOIN ent #1, ent #2

Mode: 2-D and 3-D

ent #1, ent #2: The two curves to join.

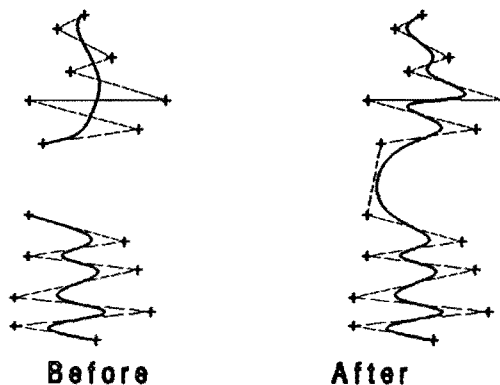
Description: Joins two curves. The resulting curve has the same type as the last curve selected. If the resulting curve is a Bézier curve, the total number of control points must be less than 12.

If the curves being joined are closed, then the end of the first curve selected is joined to the end of the first and the resulting curve is no longer closed when the two are joined. Close Curve in the Tools menu has no effect on Curve Join.

Curve Join is the exception to the rule that entities may not be joined.

The end point of ent #1 is joined to the start point of ent #2, regardless of which control point is selected. If the curves have different types, then the resultant type is the same as ent #1.

fig. (92) Join Curve



Curves

Modifying Curves



Primary

Command: CURVE MODCURVE

Modifiers:

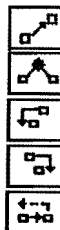
MOVECP

DELETECP

INSCPB

INSCPA

REVERSE

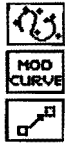


Description:

Change a selected curve by altering control points or direction of flow. In order to use this command properly, make sure GRAPHICS are turned on in the Tools menu.

These commands are similar to the control point commands in the Vector Font Editor.

Moving Control Points



Command: CURVE MODCURVE MOVECP ent #1, loc #1

Mode: 2-D and 3-D

ent #1: The control point on the curve to move.

loc #1: The location of the control point when moved.

Description: Select the control point to move and click the right mouse button. If no control point is selected, DynaCADD chooses the closest one. Select the new location for the control point and click the left mouse button. The control point is moved and the curve redrawn to suit the new location.

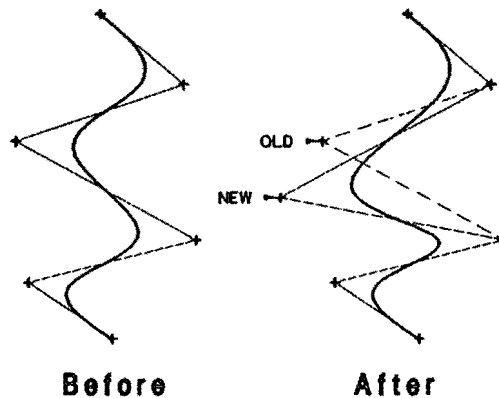


fig. (93) Move Control Point

Deleting Control Points



Command: CURVE MODCURVE DELETEDCP ent #1

Mode: 2-D and 3-D

ent #1: The control point to be deleted from the curve.
DynaCADD selects the closest control point, if none is selected.

Description: The location selected should be a control point.
DynaCADD chooses the closest control point, if one is not selected. The curve is redrawn to suit the new number of control points.

You cannot delete a control point if it reduces the number below the required minimum for that curve type.

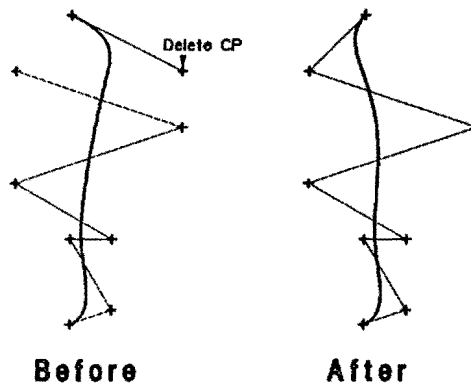


fig. (94) Delete Control Point

Inserting Control Points Before



Command: CURVE MODCURVE INSCPB ent #1, loc #1

Mode: 2-D and 3-D

ent #1: The control point on the curve, before which a new control point is to be inserted.

loc #1: The location of the inserted control point.

Description: A new control point is inserted at loc #1. This is treated as the point directly before the control point selected by ent #1. DynaCADD chooses the closest control point, if ent #1 is not selected at a control point. The curve is redrawn with the new control point added.

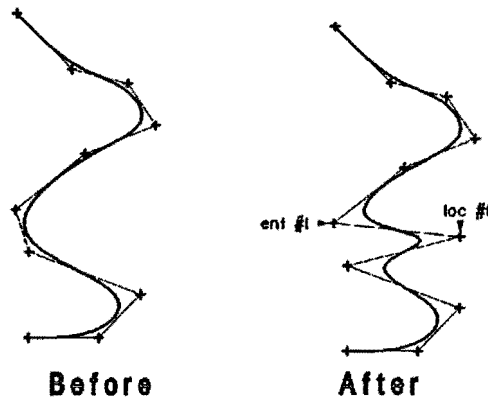


fig. (95) Insert a Control Point Before

Inserting Control Points After



MOD
CURVE



Command: CURVE MODCURVE INSCPA ent #1, loc #1

Mode: 2-D and 3-D

ent #1: The control point on the curve, after which a new control point is to be inserted.

loc #1: The location of the inserted control point.

Description: A new control point is inserted at loc #1. This is treated as the point directly after the control point selected by ent #1. DynaCADD chooses the closest control point, if ent #1 is not selected at a control point. The curve is redrawn with the new control point added.

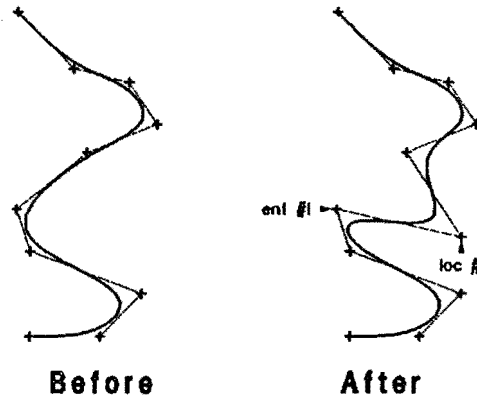


fig. (96) Insert a Control Point After

Reversing a Curve's Direction

MOD
CURVE

Command: CURVE MODCURVE REVERSE ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: Curves selected to change flow directions.

Description: This reverses the direction of flow in all selected curves.
The curve is redrawn in the new direction.

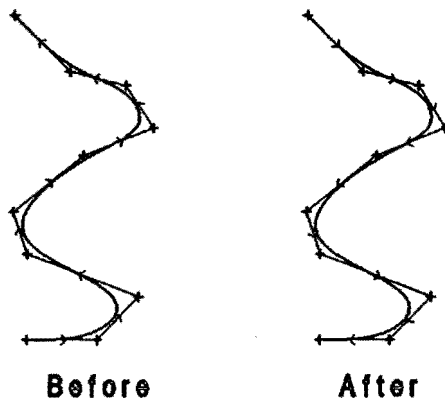


fig. (97) Reverse a Curve

Curves

—

—

—

Dimensioning ... Chapter 9

Dimensioning means measuring, and displaying, the desired dimension between locations. For example, you can measure the length of a line, the radius of a circle, or the distance between two points.

In DynaCADD, you can automatically dimension a drawing in both 2-D and 3-D modes. You can also dimension a 3-D part in 2-D, thereby removing some of the problems and complexity associated with 3-D dimensioning. There are numerous dimensioning formats and types available for use in a drawing. All dimension commands are combined under the DIMENSION icon.

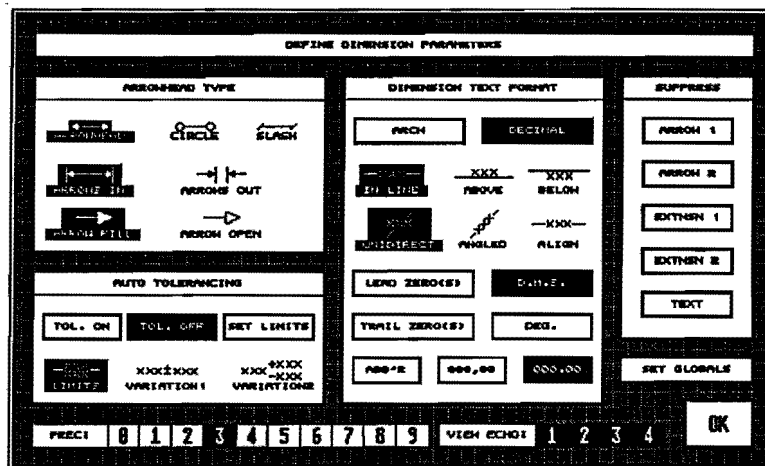


fig. (98) Dimension Parameters Dialog

All Dimension Parameters are set by selecting the Dimension option from the Set menu. This displays the Define Dimension Parameters dialog where you edit the parameters. Click the OK button to close the dialog and return to the current session.

A dimension consists of several configurable parts, including:

- arrowhead
- text
- extension
- gap
- dimension line

as explained in the following chapter.

Dimensioning

For all dimension commands, there are several general notes to take into consideration:

- You may select a font before using these commands.
- The dimension arrowheads are one third the current text height high and are equal to the current text height in length.
- The dimension text is positioned in the current font style, slant. The current character size, slant, spacing, etc. is used, but this can be changed by first using CREATE RELEASE, and then using the TRANSFORM MODIFY commands.

Although it is not a recommended practice, dimension lines and text can be altered by first releasing the dimension (CREATE RELEASE), and then selecting the appropriate transform command.

If you do not set the text width and height flags here, the values are taken from those previously entered, either here or in the INSERT TEXT command (see Chapter 7).

Commands to measure distance, perimeter and area, without generating a dimension, are found under the INFORM icon (see Chapter 14).

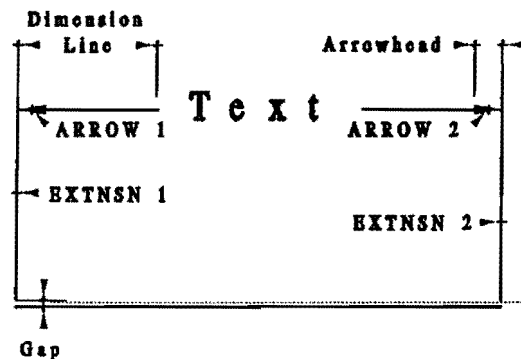


fig. (99) Dimension Line Indicating Arrowhead, Extension Lines, etc.

Arrowheads

The arrowhead is the marker used at the end of the dimension lines to indicate the dimension being measured.

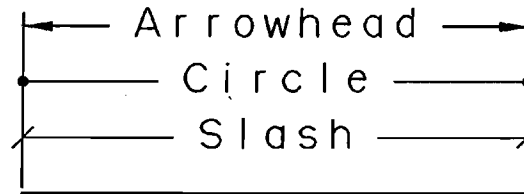


fig. (100) Sample Line Ends

Arrowhead Type

In the Arrowhead Type box are eight selectable buttons which control how the dimension line ends are displayed. The first three buttons control the type of line end used:

- Arrowhead:** Click this button to select the arrowhead as the current line end style. This style is used by most technical disciplines.
- Circle:** Click this button to select the circle as the current line end style. This is normally used in mechanical detailing to reference surfaces.
- Slash:** Click this button to select the slash as the current line end style. This is occasionally used in architectural drawings.

Dimensioning

Flipping Arrowheads

Whenever you insert a dimension, DynaCADD checks to insure that the associated text fits inside the space you provide. If the space is not large enough to accommodate the text, DynaCADD automatically flips the text outside the dimension field, to insure readability. You can also force this flipping by selecting the Arrows In and Arrows Out buttons.

Arrows In: Click this button to place all dimension arrows and associated text within the dimension field provided sufficient space is available.

Arrows Out: Click this button to place all dimension arrows and associated text outside the dimension field regardless of available space.

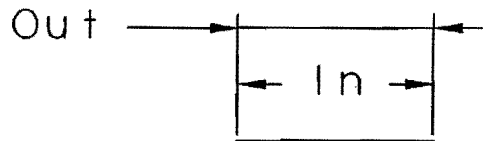


fig. (101) Arrowheads

Arrowheads Filled

You can specify whether or not the line end style currently selected is hollow or filled. These buttons have no effect on the Slash line end. In 2-D, the arrowheads are 2-D solids. In 3-D the arrowheads are filled with grouped lines to make them appear filled.

Arrow Fill: Click this button for filled (solid) arrowheads and circles in dimensions.

Arrow Open: Click this button for hollow arrowheads and circles in dimensions.

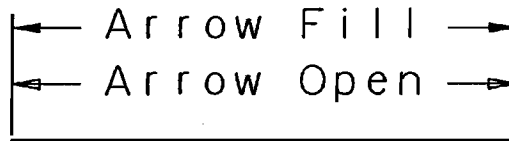


fig. (102) Filled and Open Arrowheads

Dimensioning

Auto Tolerance

In addition to automatically dimensioning a drawing, you can also instruct DynaCADD to perform auto-tolerance. That is, provide the upper and lower angular or linear tolerance values used when manufacturing a part. DynaCADD automatically determines the correct type of value (angular or linear) and inserts the values into the dimension, using the selected style (see below).

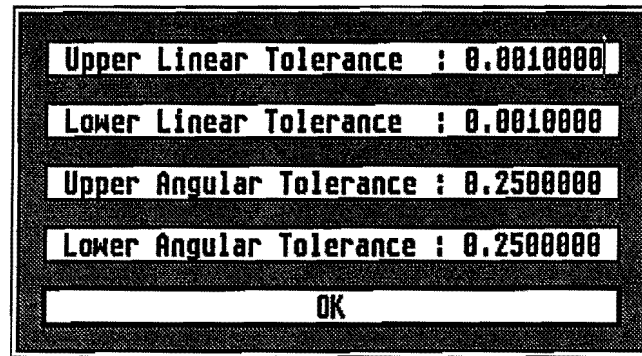


fig. (103) Set Tolerances Dialog

- Tolerance On** Click this button to enable auto tolerance. All subsequent dimensions display the tolerances in the specified style.
- Tolerance Off** Click this button to disable auto-tolerance.

Setting Limits

Click this button to display the Set Tolerances dialog. In this dialog you edit the current tolerance limits. Click the pointer on the existing value to change the text, using the normal editing keys. After you edit the existing values, click on the OK button to return to the Modify Global Dimension Parameters dialog.

- | | |
|--------------------------------|--|
| Upper Linear Tolerance | This value is always considered positive. This is the amount of variation above the basic dimension, which, when added to the basic dimension gives the largest value that is acceptable when manufacturing the part. |
| Lower Linear Tolerance | This value is always considered negative. This is the amount of variation below the basic dimension, which, when subtracted from the basic dimension, gives the lowest value that is acceptable when manufacturing the part. |
| Upper Angular Tolerance | This value is always considered positive, and must be entered in decimal degrees. This is the upper deviation permitted in an angular measurement. |
| Lower Angular Tolerance | This value is always considered negative, and must be entered in decimal degrees. The basic dimension, minus the lower angular tolerance, is the lowest limit that is acceptable when manufacturing the part. |

Selecting Tolerance Style

Choose one of the three available styles for the display of tolerance values:

- Limits** Click this button to choose the limits style of tolerance as the current style. This adds the upper tolerance value to the basic dimension and subtracts the lower tolerance value from the basic dimension, to give the tolerance limits at the top and bottom, respectively.
- Variation 1** With this variation as the current tolerance style, only one value is shown with the basic dimension, so the upper and lower tolerance values must be the same.
- Variation 2** With this variation as the current tolerance style, both the upper and lower tolerance values are displayed, one above the other.

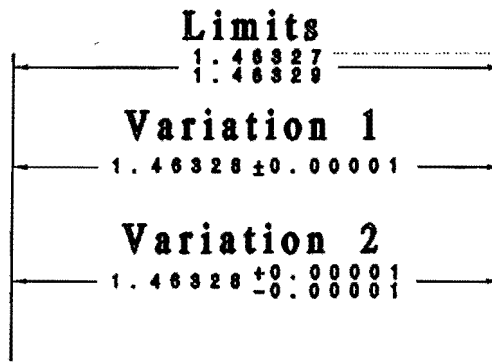


fig. (104) Examples of Tolerance Style

Dimensioning Precision

Dimensioning Precision refers to the number of characters displayed following the decimal point in dimension text. In DynaCADD you have the option to select between zero and nine characters after the decimal. To select the precision desired, click the pointer over the appropriate buttons labeled 0 to 9. Internally, DynaCADD uses 16 digits of precision.

Dimension Text Format

You can format the text associated with a dimension in numerous ways, to tailor the text to your particular needs:

Text Style

- | | |
|----------------------|--|
| Architectural | Architectural dimensioning may involve the use of fractions and the separation of the text into feet and inches. Click this button to separate all dimension text into feet, inches and fractions of inches. DynaCADD automatically rounds numbers to the nearest 1/64th inch, and selects the lowest denominator for the fractional part of the text. |
| Decimal | Click this button to display all dimension text as a floating point number, with the number of characters following the decimal point defined by the current precision. |

Text Placement

Text placement refers to the position the text occupies in the dimension:

- | | |
|---------------|---|
| Inline | Click this button to insert all text inside the actual dimension. The dimension line is divided and trimmed a sufficient distance in order to insure readability. |
| Above | Click this button to place all text above the dimension line. |
| Below | Click this button to place all text below the dimension line. |

Dimensioning

Text Rotation

Text rotation refers to the rotation of the text relative to the dimension line. There are three styles available:

- | | |
|------------------|--|
| Unidirect | Click this button to place all text along the X-axis with no rotation applied. |
| Angled | Click this button to place all text at a rotation equal to that of the dimension line. |
| Aligned | Click this button to place all text at either 0 or 90 degrees rotation, depending on which provides the least deviation from the dimension line. |
| Note: | These changes do not affect existing dimension text, only new text inserted after the button is clicked. |

Text Abbreviations

You can further format text by automatically adding the current unit abbreviation and stripping leading and/or trailing zeros from the text.

- | | |
|-------------------------------|--|
| Strip Leading Zero(s) | Click this button to strip all leading zeros from the dimension text. |
| Strip Trailing Zero(s) | Click this button to strip all trailing zeros from the dimension text. |
| Add Abbreviation | Click this button to add the abbreviation of the current unit type to the text. |
| Deg/D.M.S. | Click one of these buttons to determine whether the angular dimensions will be shown in degrees and decimal places, or degrees/minutes/seconds format. |

Decimal Conventions

There are two decimal conventions supported by DynaCADD that specify which character represents the decimal point:

- | | |
|---------------|---|
| 000,00 | Click this button to use the comma to represent the decimal position (European style). |
| 000.00 | Click this button to use the period to represent the decimal position (North American style). |

Partial Suppression of Dimensions

There are five buttons provided to suppress individual portions of a dimension:

- | | |
|------------------|---|
| Arrow 1: | Any inserted dimension does not display the arrowhead of the first selected location. |
| Arrow 2: | Any inserted dimension does not display the arrowhead of the second selected location. |
| Extnsn 1: | Any inserted dimension does not display the extension line of the first selected location. |
| Extnsn 2: | Any inserted dimension does not display the extension line of the second selected location. |
| Text: | Any inserted dimension does not display the associated text. |

Echoing Dimensions in Active Views

Normally, all inserted 3-D dimensions are displayed in all active views. Since this is rarely required, you can specify in which view(s) to echo the dimension. This way, you can use 3-D dimensioning without crowding all active views with unnecessary information. The four buttons labeled 1 to 4 represent the four available views, respectively. Views are numbered in the order of their creation. If you click a numbered button, the corresponding view echoes any 3-D dimension inserted.

Dimensioning

Setting Globals

Click here to set the global parameters for dimension entities. These parameters affect all dimensions.

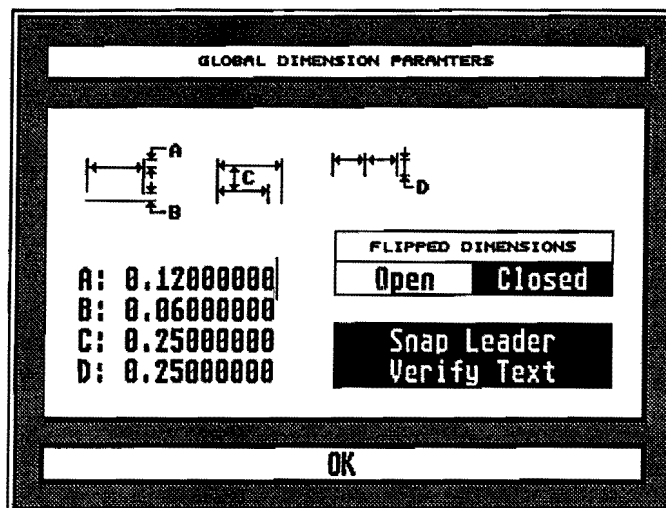


fig. (105) Global Dimension Parameters Dialog

- A** This determines the upper extension, the distance between the dimension line and the top of the perpendicular extension line.
- B** This is the gap, the distance between the extension and the entity or location being measured.
- C** This is the distance between successive (parallel) vertical or horizontal dimensions when using DIMHBASE and DIMVBASE commands.
- D** This is length of the extension line, from the dimension line to the bottom of the extension, when using CHCHAIN and CVCHAIN commands. Parameter A (above) is used for the upper extension.

Flipped Dimensions

This determines if the dimension is displayed in flipped dimensions (dimensions that are displayed outside the dimension area) appear open (no line across the dimension) or closed (line extending across the dimension).

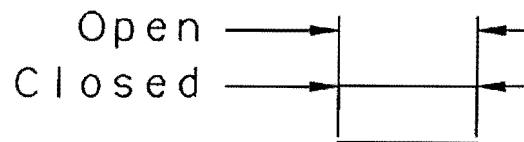


fig. (106) Opened and Closed Flipped Dimensions

Snap Leader

This determines if the last line of the leader is automatically snapped to the horizontal grid. The leader text is always aligned with the horizontal grid.

Verify Text

With this on, the dimension text must be verified for each dimension (but not leaders). You can edit the dimension text in the dialog. Press [Return] to accept the text on the line.

This gives you the opportunity to edit the text and, if necessary, change the dimension value.

Dimensioning

Dimensions



**Primary
Command:**

DIMENSION

Modifiers:

DIMLOC



DIMLINE



DIMHORI



DIMVERT



DIMRAD



DIMDIA



DIMHCHAIN



DIMVCHAIN



CHCHAIN



CVCHAIN



DIMHBASE



DIMVBASE



DIMCLINE



DIMCCLINE



DIMANGLE



DIMLEADER



TEXTWID (flag)



TEXTHEI (flag)



TEXTSLA (flag)



Flags:

DynaCADD assumes that the value given is either in inches or millimeters exclusively, depending on the currently selected unit type, English or Metric, respectively.



TEXTWID Defines the width of a single character box.



TEXTHEI Defines the height of a single character box, based on the distance from the baseline to the top of the uppercase "A".



TEXTSLA Defines the slant (-60 to $+60$ degrees) applied to each character in the text. Slanted text such as italic, slants to the right 10 to 12 degrees, entered as negative value.

Description: Insert dimension lines and text at the selected locations or entities.

Dimensioning

Dimensioning Locations



Command: DIMENSION DIMLOC (flags) loc #1, loc #2, loc #3

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #2: Locations which specify the distance to dimension. The dimension is inserted at a angle equal to that of a vector defined by the two locations.

loc #3: Specifies the distance from the dimensioned locations to place the dimension line and text. This distance is the normal distance to an imaginary line defined by location #1 and location #2.

Description: The distance between location #1 and location #2 is dimensioned. The dimension text is positioned at location #3.

Note: If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #3.

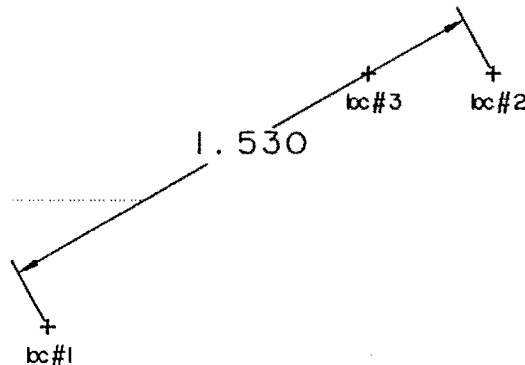


fig. (107) Dimension Locations

Dimensioning Lines



- Command:** DIMENSION DIMLINE (flags) ent #1, loc #1
- Mode:** 2-D and 3-D
- Flags:** See primary DIMENSION command.
- ent #1:** The line to dimension. In 2-D mode, you can select a 3-D line to dimension, but the 3-D line ends are automatically converted to 2-D locations and are dimensioned accordingly. The dimension are inserted at a angle equal to that of a vector defined by the two line end points.
- loc #1:** The distance from the dimensioned locations to place the dimension line and text. This distance is the normal distance to the line.
- Description:** The distance between the line end points is dimensioned and the dimension text is positioned at location #1.
- Note:** If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #1.

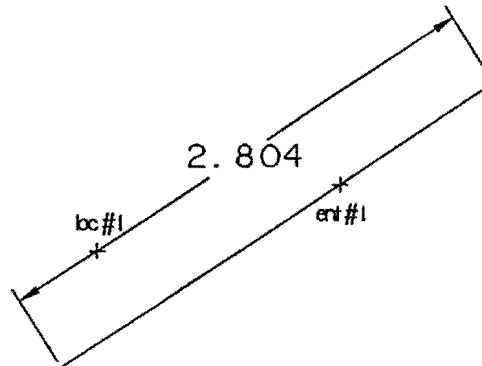


fig. (108) Dimensioning a Line

Dimensioning

Dimensioning Horizontal Distances



Command: DIMENSION DIMHORI (flags) loc #1, loc #2, loc #3

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #2: Locations that specify the horizontal distance to dimension. The dimension is inserted horizontally between location #1 and location #2.

loc #3: The location where the dimension line and text are placed.

Description: The horizontal distance between location #1 and location #2 is dimensioned. The dimension text is positioned at location #3.

Note: If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #3.

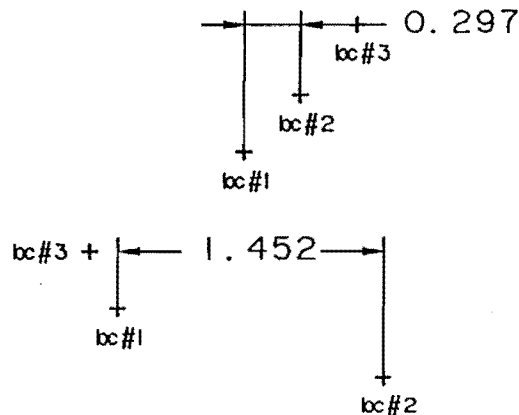


fig. (109) Dimension Horizontal Distance

Dimensioning Vertical Distances



- Command:** DIMENSION DIMVERT (flags) loc #1, loc #2, loc #3
- Mode:** 2-D and 3-D
- Flags:** See primary DIMENSION command.
- loc #1, loc #2:** Locations that specify the vertical distance to dimension. The dimension is inserted vertically between location #1 and location #2.
- loc #3:** The location where the dimension line and text are placed.
- Description:** The vertical distance between location #1 and location #2 is dimensioned. The dimension text is positioned at location #3.
- Note:** If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #3.

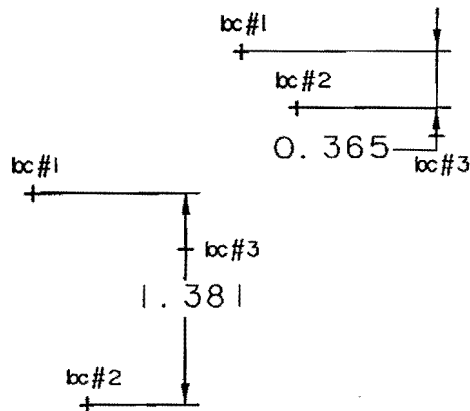


fig. (110) Dimension Vertical Distance

Dimensioning Radial



Command: DIMENSION DIMRAD (flags) ent #1, loc #1

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

ent #1: Circle, arc or fillet to dimension. In 2-D mode, you can select a 3-D entity to dimension, provided that the circle, arc or fillet image conversion from 3-D to 2-D does not result in an elliptically shaped entity.

loc #1: The location where the dimension line and text are placed, relative to the origin of the selected entity.

Description: The radius of the selected entity is dimensioned. The dimension text is positioned inside the circle radius if location #1 is inside and there is sufficient room for the text, otherwise the text and arrowhead are automatically flipped outside the circle.

Note: If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #1.

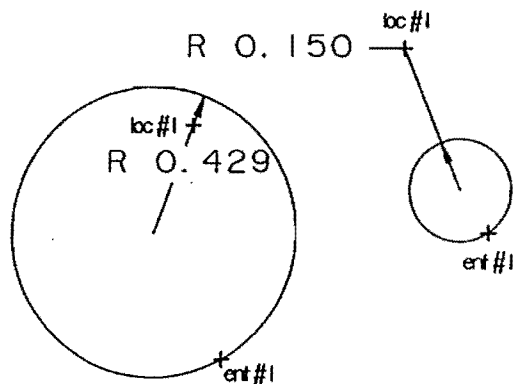


fig. (III) Dimension Radius

Dimensioning Diameters



Command: DIMENSION DIMDIA (flags) ent #1, loc #1

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

ent #1: Circle, arc or fillet to dimension. In 2-D mode, you can select a 3-D entity to dimension, provided that the circle, arc or fillet image conversion from 3-D to 2-D does not result in an elliptically shaped entity.

loc #1: The location where the dimension line and text are placed, relative to the origin of the selected entity.

Description: The diameter of the selected entity is dimensioned. The dimension text is positioned inside the circle radius if location #1 is inside and there is sufficient room for the text, otherwise the text and arrowhead are automatically flipped outside the circle.

Note: If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #1.

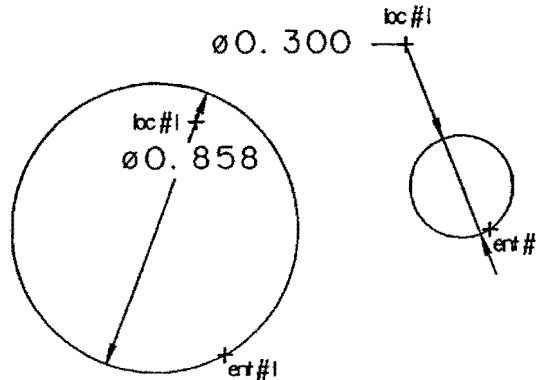


fig. (112) Dimensioning Diameters

Dimensioning

Horizontal Chain Dimensioning



Command: DIMENSION DIMHCHAIN (flags) loc #1...loc #n

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #n-1: Locations specifying the horizontal distance(s) to dimension. The dimension is inserted horizontally between location #1 and location #2. If more than three locations are selected, the horizontal distances between all selected locations are dimensioned in order of their selection.

loc #n: The last location selected, where the dimension line and text are placed for all dimensions.

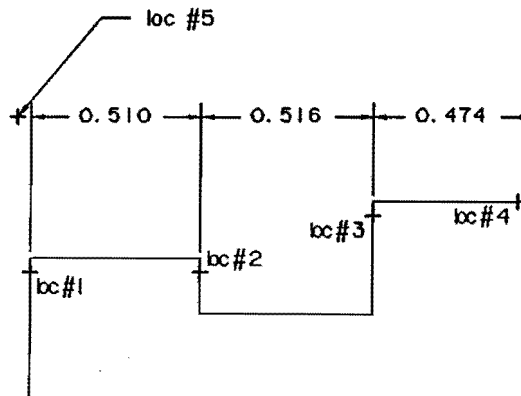


fig. (113) Horizontal Chain Dimensioning

Description:

Activate the command by clicking the right mouse button, or by pressing [;].

The horizontal distance between location #1 and location #2 is dimensioned. If more than three locations are selected, the horizontal distance between location #2 and location #3 is then dimensioned. This process continues until the horizontal distance between location #n-2 and location #n-1 has been dimensioned. The dimension text is positioned at the last selected location.

The extension lines extend from the baseline of the dimension text to the locations specified from loc #1 to loc # n-1. See CHCHAIN, below.

Note:

If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #n.

Dimensioning

Vertical Chain Dimensioning



Command: DIMENSION DIMVCHAIN (flags) loc #1...loc #n



Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #n-1: Locations specifying the vertical distance(s) to dimension. The dimension is inserted vertically between location #1 and location #2. If more than three locations are selected, the vertical distances between all selected locations are dimensioned in order of their selection.

loc #n: The location where the dimension line and text are placed, for all dimensions. This is the last location selected.

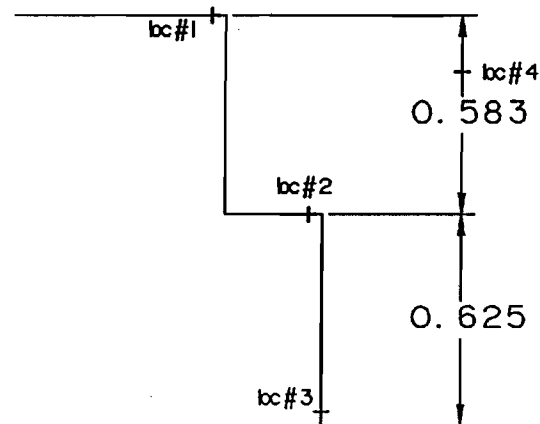


fig. (114) Vertical Chain Dimensioning

Description:

Activate the command by clicking the right mouse button, or by pressing [;].

The vertical distance between location #1 and location #2 is dimensioned. If more than three locations are selected, the vertical distance between location #2 and location #3 is then dimensioned. This process continues until the vertical distance between location #n-2 and location #n-1 has been dimensioned. The dimension text is positioned at the last selected location.

The extension lines extend from the baseline of the dimension text to the locations specified from loc #1 to loc # n-1. See CVCHAIN, located on the previous page.

Note:

If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #n.

Dimensioning

Constant Horizontal Chain Dimensioning



Command: DIMENSION CHCHAIN (flags) loc #1...loc #n



Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #n-1: Locations specifying the horizontal distance(s) to dimension. The dimension is inserted horizontally between location #n-1 and location #n. If more than three locations are selected, the horizontal distances between all selected locations are dimensioned in order of their selection.

loc #n: The location where the dimension line and the baseline of the dimension text are placed, for all dimensions. This is the last location selected.

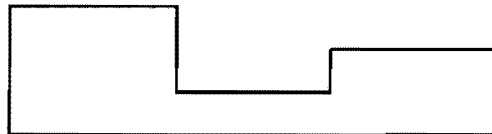
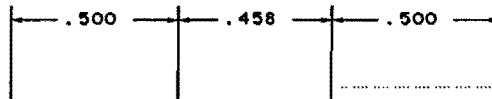


fig. (115) Constant Horizontal Chain Dimensioning

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The horizontal distance between location #1 and location #2 is dimensioned. If more than three locations are selected, the horizontal distance between location #2 and location #3 is then dimensioned. This process continues until the horizontal distance between location #n-2 and location #n-1 has been dimensioned. The baseline of the dimension text is positioned at the last selected location.

The difference between this command and DIMHCHAIN is that the extension lines are always a constant length, as set in the global parameters of the Dimension Parameters dialog.

Note: If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #n.

Dimensioning

Constant Vertical Chain Dimensioning



Command: DIMENSION CVCHAIN (flags) loc #1...loc #n

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #n-1: Locations specifying the vertical distance(s) to dimension. The dimension is inserted vertically between location #1 and location #2. If more than three locations are selected, the vertical distances between all selected locations are dimensioned in order of their selection.

loc #n: The location where the dimension line and text are placed, for all dimensions. This is the last location selected.

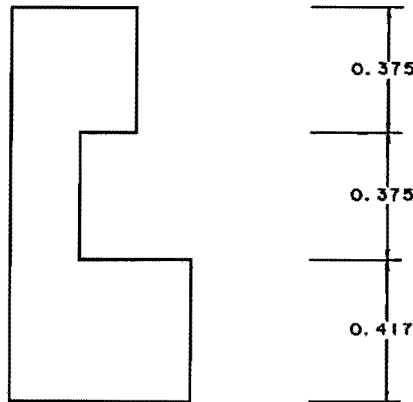


fig. (116) Constant Vertical Chain Dimensioning

- Description:** Activate the command by clicking the right mouse button, or by pressing [;].
- The vertical distance between location #1 and location #2 is dimensioned. If more than three locations are selected, the vertical distance between location #2 and location #3 is then dimensioned. This process continues until the vertical distance between location #n-2 and location #n-1 has been dimensioned. The dimension text is positioned at the last selected location.
- The difference between this command and DIMVCHAIN is that the extension lines are always a constant length, as set in the global parameters of the Dimension Parameters dialog.
- Note:** If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #n.

Dimensioning

Horizontal Baseline Dimensioning



Command: DIMENSION DIMHBASE (flags) loc #1...loc #n

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #n-1: Locations specifying the horizontal distance(s) to dimension. The dimension is inserted horizontally between location #1 and location #2. If more than three locations are selected, the horizontal distances between all selected locations are dimensioned in order of their selection.

loc #n: The location where the dimension line and text are placed, for the initial dimension. This is the last location selected.

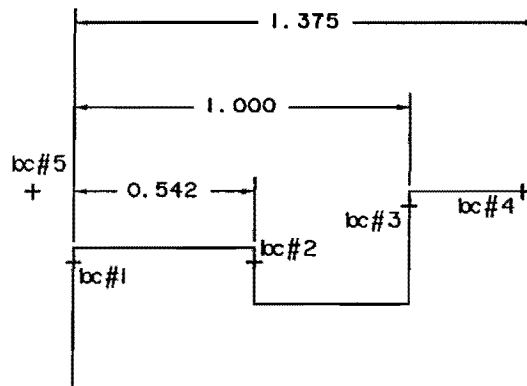


fig. (117) Horizontal Baseline Dimensioning

- Description:** Activate the command by clicking the right mouse button, or by pressing [;].
- The horizontal distance between location #1 and location #2 is dimensioned. If more than three locations are selected, the horizontal distance between location #2 and location #3 is then dimensioned. This process continues until the horizontal distance between location #n-2 and location #n-1 has been dimensioned. The dimension text is positioned at the last selected location.
- Note:** If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #n.

Dimensioning

Vertical Baseline Dimensioning



Command: DIMENSION DIMVBASE (flags) loc #1...loc #n

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

loc #1, loc #n-1: Locations specifying the vertical distance(s) to dimension. The dimension is inserted vertically between location #1 and location #2. If more than three locations are selected, the vertical distances between all selected locations are dimensioned in order of their selection.

loc #n: The location where the dimension line and text are placed, for the initial dimension. This is the last location selected.

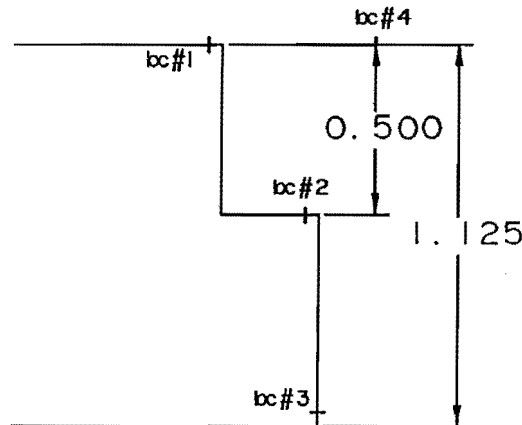


fig. (118) Vertical Baseline Dimensioning

- Description:** Activate the command by clicking the right mouse button, or by pressing [;].
- The vertical distance between location #1 and location #2 is dimensioned. If more than three locations are selected, the vertical distance between location #2 and location #3 is then dimensioned. This process continues until the vertical distance between location #n-2 and location #n-1 has been dimensioned. The dimension text is positioned at the last selected location.
- Note:** If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #n.

Dimensioning

Inserting Center Lines



Command: DIMENSION DIMCLINE: ent #1

Mode: 2-D and 3-D

ent #1: Circle, arc, fillet, ellipse or elliptical arc. In 2-D mode, you can select a 3-D entity to dimension, however, each center line is inserted on the 3-D plane on which the corresponding entity lies.

Description: A center line is inserted at the origin, and extends 1/8 inch past the circumference of the selected entity. The line style is defined as SYS_CENTER, consistent with the standard center line. See Chapter 6.

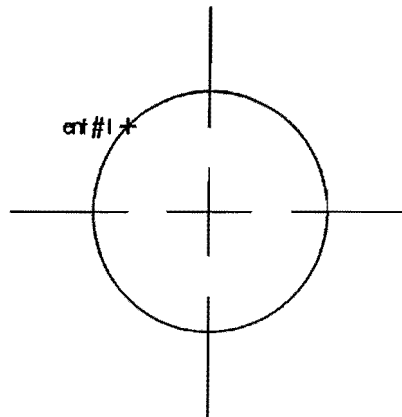


fig. (119) Center Lines

Inserting Center Lines on Base Circle Diameters



Command: DIMENSION DIMCCLINE: loc #1, ent #1...ent #n

Mode: 2-D and 3-D

loc #1: The location defining the origin of the base circle.

ent #1...ent #n: One or more circles, arcs, or fillets, situated on the circumference of the base circle. In 2-D mode, you can select a 3-D entity to dimension.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

A circular center line is placed with its origin at location #1. The radius of the circular center line is the distance from location #1 to the origin of the first selected location. At each of the selected entities, a center line is inserted through the origin, at an angle equal to that between location #1 and the origin of the corresponding entity.

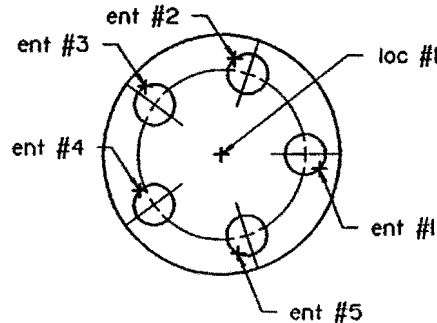


fig. (120) Circular Center Lines

Dimensioning Angles



Command: DIMENSION DIMANGLE (flags) ent #1, ent #2, loc #1

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

ent #1, ent #2: Two lines for which the angle is measured. In 2-D mode, you can select 3-D lines to dimension, in which case an image of the 3-D line is automatically converted to a corresponding 2-D line.

loc #1: The location of the dimension arc and text.

Description: The angle between the two selected lines is dimensioned. If location #1 is between the two lines, and there is sufficient room to insure readability, the text is placed within the angle, otherwise the text is placed outside the line, closest to location #1.

DynaCADD automatically dimensions the inside, or smaller, angle.

Note: If the Arrows Out button is selected or there is insufficient space to place the dimension text inside the dimension field, the text is flipped outside the dimension closest to location #1.

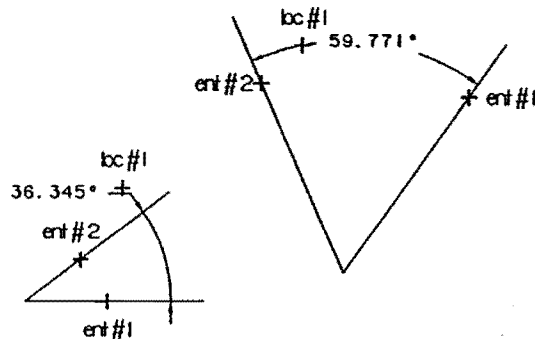


fig. (121) Dimensioning Angles

Leaders



Command: DIMENSION DIMLEADER (flags) str: loc #1...loc #n

Mode: 2-D and 3-D

Flags: See primary DIMENSION command.

str: After you select the LEADER icon, type the required text into the text editor and press [Enter]. If a leader line is required with no text, simply press [Enter] without entering any text.

loc #1, loc #n: Locations specifying the leader line ends. The last line of the leader is inserted horizontally.

Description: A leader is a user defined text description or label, connected between loc #1 and loc #n with an arrow and a series of lines. If Snap Leader is on in the Dimension Parameters dialog, then the last line in the leader has any vertical deviation reduced to zero and is snapped to the horizontal axis. The leader text, if any, is inserted at location #n, always along the horizontal axis.

In order to modify leader text, you must first release it (CREATE RELEASE, see Chapter 12). Then you can use the Transform commands to alter the text or text attributes (see TRANSFORM, Chapter 10).

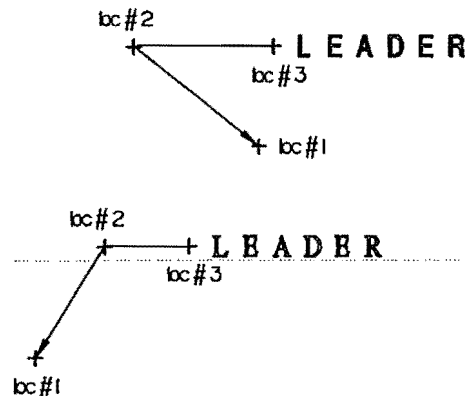


fig. (122) Leaders

Dimensioning

Transformation ... Chapter 10

Transformation commands modify, move, rotate, scale, or delete entities. DynaCADD has a large array of commands specifically designed to allow you to modify entities. You can also use some of these commands to create new entities by copying the original entities to new locations. Mastering these commands greatly decreases design time.

The TRANCOPY (transform copy) icon, located on the left side of the top icon strip, affects how many of these commands act upon the selected entities. Which commands are affected is noted in the descriptions. If TRANCOPY is on, the command affects an image of the original entities and the original entities are left undisturbed. If TRANCOPY is off, the command affects the original entities.

To show what entities are valid for a particular command, click on the Entity Filter icon (FILTER). All valid entities for the command are displayed in the dialog. Any invalid entities are disabled.

All transformation commands are combined under the TRANSFORM icon.

Deleting Entities

**Primary****Command:** TRANSFORM DELETE**Modifiers:** None**Command:** TRANSFORM DELETE: ent #1...ent #n**Mode:** 2-D & 3-D**ent #1...ent #n:** Any number of entities to delete.

Description: DELETE removes all selected entities from the drawing. As with all transformation commands, you may only select those entities created in the current mode (2-D or 3-D), for deletion.

This is the same delete command as in the INSERT icon pad.

Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities are permanently removed from the drawing. This command can be undone, as long as no other command is activated before [Undo] is pressed.

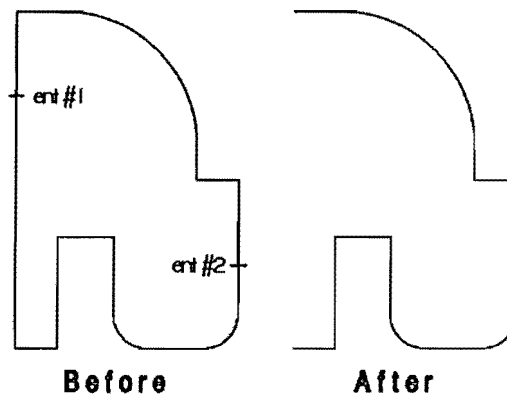


fig. (123) Deleting Entities

Trimming Entities


Primary
Command: TRANSFORM TRIM

Modifiers:

TRIMONE



TRIMCORN



TRIMINTOF



TRIMMULTI



TRIMINC



TRIMDIST



Description: TRIM shortens or lengthens an entity, depending on where, in relation to the current length, you select as a new entity end. Trim works on lines, arcs, fillets and elliptical arcs. You can only trim circles and ellipses after dividing them into two or more arcs (see Dividing Entities, below).

There are six types of trim:

ONE Trim one selected entity.

CORN Trim two entities to their intersection.

INTOF Trim one entity to an intersection.

MULTI Trim several entities at once.

INC Trim an entity by a specified increment.

DIST Trim an entity to an absolute distance.

Trim responds to the location and entity ends selected, according to the type of trim being performed. The entity end closest to the location selected is used in a trim. Be precise when selecting locations and entities to trim.

Trimming One Entity



Command: TRANSFORM TRIM TRIMONE: ent #1, loc #1



Mode: 2-D and 3-D



ent #1: The entity to trim.

loc #1: The location of the trim. The trim is applied to the entity end, closest to location #1, according to the distance between the end and loc #1.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Select the end to trim and a new location. DynaCADD intersects the line or arc perpendicular to the point chosen and performs the trim. The end point of the selected entity closest to location #1 is appropriately lengthened or shortened to that location.

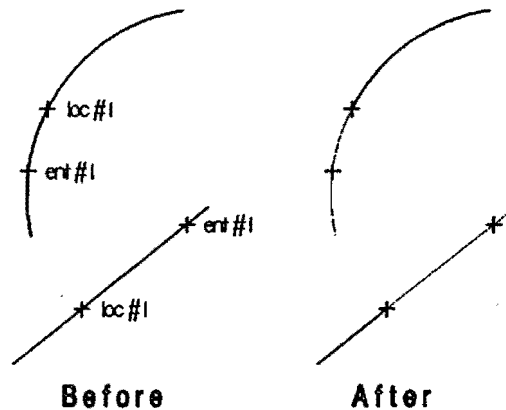


fig. (124) Trim One Entity

Trimming at the Corner of Two Entities



Command: TRANSFORM TRIM TRIMCORN: ent #1, ent # 2

Mode: 2-D and 3-D

ent #1, ent #2: The two entities to trim. The entities should intersect or be located in such a way that they will intersect when trimmed. Parallel entities cannot intersect.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The trim is performed according to the same rules as when inserting fillets. The selection point closest to the intersection is used. The end points of the selected entities closest to the selection point are appropriately lengthened or shortened to the intersection location.

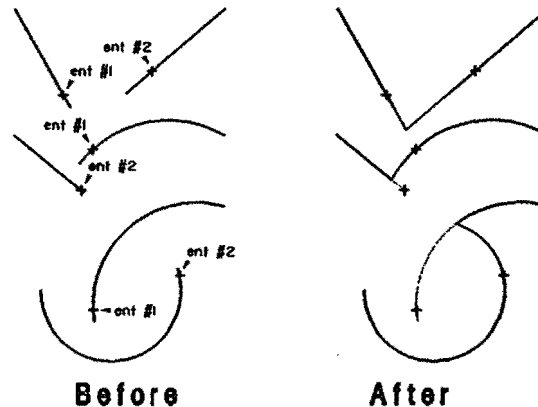


fig. (125) Trim at the Corner of Two Entities

Trimming One Entity to an Intersection



Command: TRANSFORM TRIM TRIMINTOF: ent #1, ent #2



Mode: 2-D and 3-D



ent #1: Entity to be trimmed.

ent #2: Entity to trim to. The first entity should intersect the second entity or be located in such a way that will intersect when trimmed. Parallel entities cannot intersect.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The trim is performed according to the same rules as when inserting fillets. The selection point closest to the intersection is used. The end points of the selected entities closest to the selection point are appropriately lengthened or shortened to the intersection location.

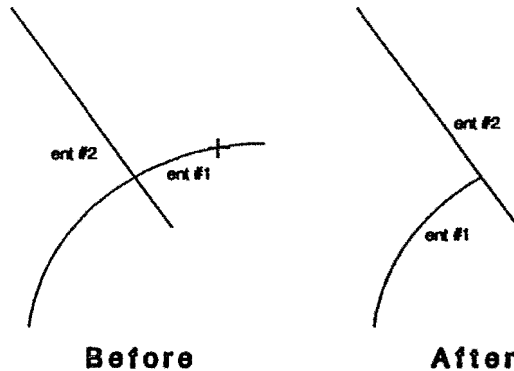
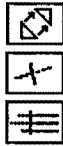


fig. (I26) Trim One Entity to its Intersection with Another

Trimming Several Entities



Command: TRANSFORM TRIM TRIMMULTI ent #1...ent #n,
loc #1

Mode: 2-D and 3-D

ent #1...ent #n: The entities to trim. The ends closest to the selected entities are trimmed to location #1.

loc #1: The location of the trim. All entities are trimmed perpendicular to the line being trimmed, from the trimming point (location #1). The trimming is applied to the selected end of the entity end.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The end points of the selected entities closest to location #1 are appropriately lengthened or shortened to a position perpendicular to location #1.

If the entity being trimmed is angled to the axes, then the trim is perpendicular to the entity, not the axes.

This is useful for aligning entity ends with a particular location.

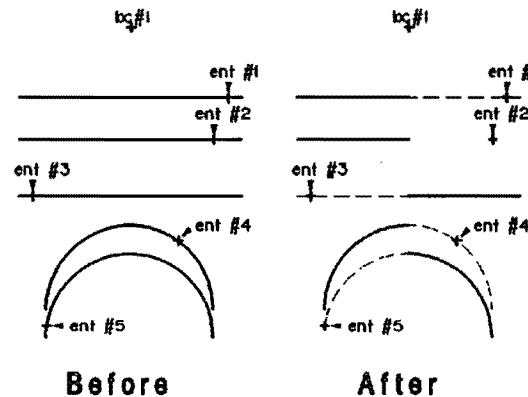


fig. (127) Trim Several Entities to a Specific Location

Transformation

Trimming One Entity by an Increment



Command: TRANSFORM TRIM TRIMINC (inc) ent #1



Mode: 2-D and 3-D



inc: Absolute amount to trim entity; enter a positive value to lengthen, a negative value to shorten.

ent #1: The entity to trim.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Enter the amount to trim into the pop up calculator. The end point of the selected entity closest to the selection point is lengthened by that amount if the value is positive, or shortened if the value is negative.

Distances for arcs are calculated along their circumference.

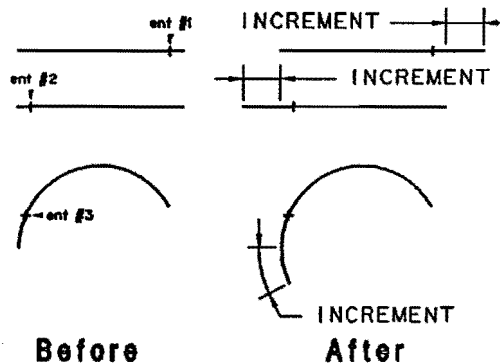


fig. (128) Trim by an Increment

Trimming One Entity to an Absolute Distance



Command: TRANSFORM TRIM TRIMDIST (dist) ent #1

Mode: 2-D and 3-D

dist: The absolute length of the entity. Must be a positive number.

ent #1: The entity to trim.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Enter the absolute length of the entity into the pop up calculator. The value can only be positive. The end point of the selected entity closest to location #1 is appropriately lengthened or shortened so that the entity is the entered length, when trimmed.

Distances for arcs are calculated along their circumference.

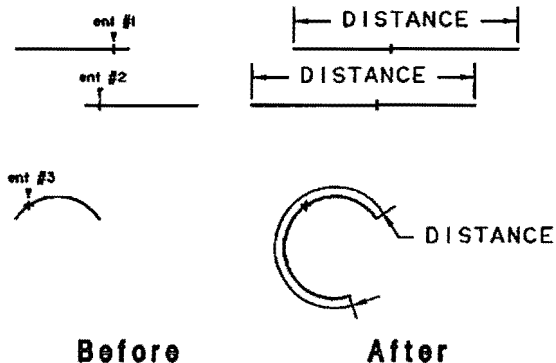


fig. (129) Trim to an Absolute Distance

Transformation

Dividing Entities



Primary

Command: TRANSFORM DIVIDE

Modifiers: DIVLOC



DIVNDIV



Description: DIVIDE breaks an entity into two or more separate entities. Dividing a circle creates two or more arcs. Dividing an ellipse creates two or more elliptical arcs. To divide Bézier and B-spline curves, use the CURVE commands described in Chapter 8.

Dividing Entities by Location

Command: TRANSFORM DIVIDE DIVLOC: ent #1, loc #1

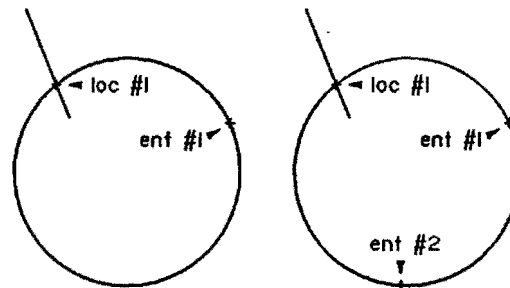
Mode: 2-D and 3-D

ent #1: Line, circle, arc, fillet, ellipse or elliptical arc to divide.

loc #1: The location where the entity is to divide.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The selected entity is divided at location #1 and broken into two separate entities.



Before

After

fig. (130) Dividing an Entity by Location

Dividing Entities Into Equal Divisions



Command: TRANSFORM DIVIDE DIVNDIV (numdiv): ent #1

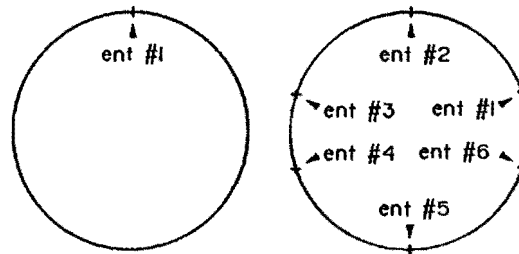
Mode: 2-D and 3-D

numdiv: The number of equal divisions into which the selected entity is broken.

ent #1: Line, circle, arc, fillet, ellipse or elliptical arc to divide.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The selected entity is divided into “numdiv” divisions of equal size, which creates the same number of separate entities. Line divisions are based on equal lengths, and all other entities are broken into entities of equal angular deviations.



Before

After

fig. (131) Divide an Entity into Equal Divisions

Translating Entities



Primary

Command:

TRANSFORM TRANSLATE

Modifiers:

TRANMOVE



TRANCOPY



TRANMCOPY



TRANSFACE (flag)



Flag:

TRANSFACE Generate 3-D faces from the original entities to their copies. This only applies to 3-D entities.

Description:

Move, copy or create multiple copies of the selected entities.

There are three types of translate:

MOVE

Moves the selected entities.

COPY

Copies the selected entities.

MCOPY

Creates more than one copy of the selected entities.

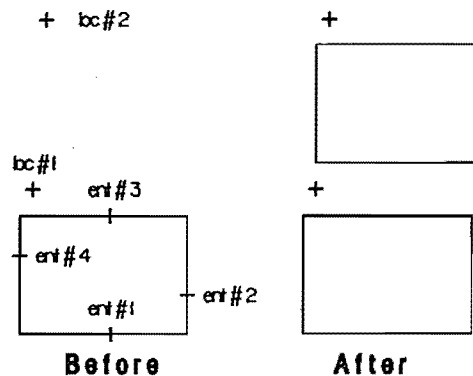
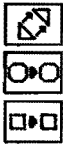


fig. (132) Translating Entities

Transformation

Moving Entities



Command: TRANSFORM TRANSLATE TRANMOVE:

ent #1...ent #n, [:] loc #1, loc #2

Mode: 2-D and 3-D

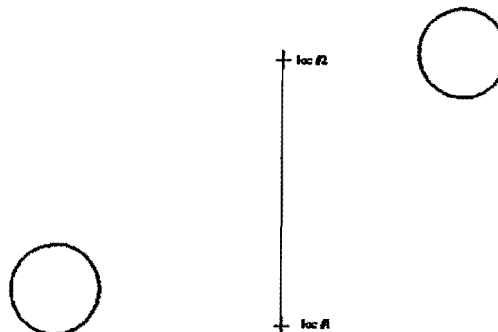
ent #1...ent #n: One or more entities to move. You may select all entity types.

loc #1, loc #2: These two locations define a vector along which the entities are translated. Imagine the first location as the "from" position and the second location as the "to" position.

Description: Activate the command by clicking the right mouse button, or by pressing [:].

The TRANMOVE command moves a single entity or group of selected entities along the vector defined by location #1 and location #2, from the former to the latter.

Note: The TRANCOPY icon found above the drawing area does not affect this command.



Before

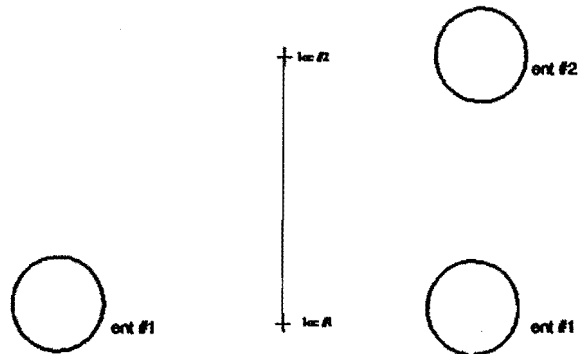
After

fig. (133) Moving Entities

Copying Entities



- Command:** TRANSFORM TRANSLATE TRANCOPY (flag):
ent #1...ent #n, [:] loc #1, loc #2
- Mode:** 2-D and 3-D
- Flag:** See Primary TRANSFORM TRANSLATE command.
- ent #1...ent #n:** One or more entities to copy. You may select all entity types.
- loc #1, loc #2:** These two locations define a vector along which the entities are translated. Imagine the first location as the "from" position and the second location as the "to" position.
- Description:** Activate the command by clicking the right mouse button, or by pressing [:].
- The TRANCOPY command copies a single entity or group of selected entities along the vector defined by location #1 and location #2, from the former to the latter.
- Note:** The TRANCOPY icon found above the drawing area



Before

After

fig. (I34) Copying Entities

Copying Entities to Multiple Locations



Command: TRANSFORM TRANSLATE TRANMCOPY (flag):
ent #1...ent #n, [;] loc #1, loc #2 [;]...loc #n [;]

Mode: 2-D and 3-D

Flag: See Primary TRANSFORM TRANSLATE command.

ent #1...ent #n: One or more entities to copy. You may select all entity types.

loc #1: A location relative to the selected entities.

loc #2...loc #n: These locations define a vector along which the entities are translated. Imagine the location #1 as the “from” position and all other locations as the “to” position.

Description: The TRANMCOPY command copies a single entity or group of selected entities along the vector defined by location #1 and all other locations, from the former to the latter.

Begin this command by first selecting the entities to be copied and clicking the right mouse button, or by pressing [;] to end entity selection. Now select the original location on the drawing page with the left mouse button.

Activate this command by selecting the new location on the drawing page and clicking the right mouse button, or pressing [;] to make a copy of the selected entities.

Create another copy by selecting another new location and clicking the right mouse button. Repeat this process for all successive copies.

Note: The TRANCOPY icon found at the top of the drawing page does not affect this command.

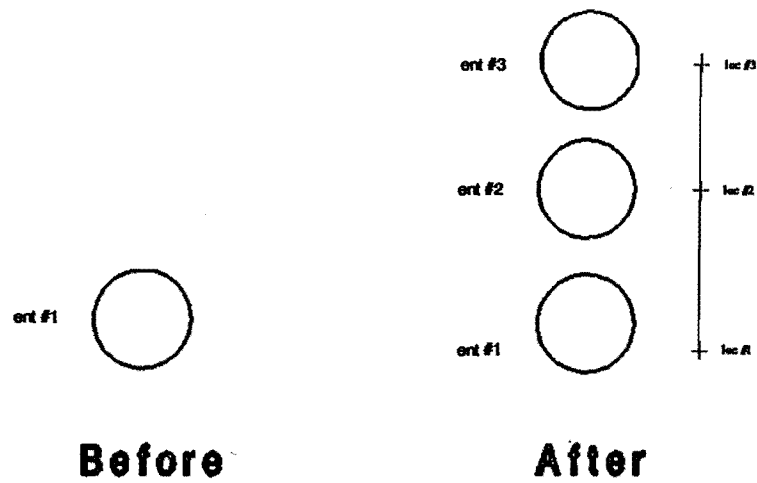


fig. (135) Copying Entities to Multiple Locations

Transformation

Rotating Entities

**Primary****Command:** TRANSFORM ROTATE**Modifiers:** ROTANG (flag) ROTNCOPY (flag) 

Flags:

ROTANG The angle of rotation (-360 to +360 degrees) by which the selected entities are rotated. The default is 90 degrees.

ROTNCOPY The number of repetitions to perform by the ROTATE command. The entities are incremented by ROTANG degrees, for each repetition. The default is one. A copy is made for each rotation if the value is greater than one.

Description: This command rotates a selected group of entities about a definable origin. You can also create circular arrays using this command (for other arrays, see TRANSFORM ARRAY, located on the previous page).

Rotating Entities About a Selected Location



- Command:** TRANSFORM ROTATE (flags) ent #1...ent #n, [;]
loc #1
- Mode:** 2-D and 3-D
- Flags:** See primary TRANSFORM ROTATE command.
- ent #1...ent #n:** One or more entities to rotate. You may select all entity types.
- loc #1:** The origin about which all the selected entities are rotated. Imagine this as the center of a circle about which the entities rotate.
- Description:** Activate the command by clicking the right mouse button, or by pressing [;].
- All selected entities are rotated by ROTANG degrees about location #1. The command repeats the rotation ROTNCOPY times, incrementing the angle with each repetition and making a copy each time. In 3-D mode, the entities are rotated on the current plane defined by the active GCP.
- Note:** The TRANCOPY modifier in the Tools menu affects this command. However, if the number of repetitions specified by ROTNCOPY is greater than one, DynaCADD automatically applies a Copy regardless of the state of TRANCOPY.

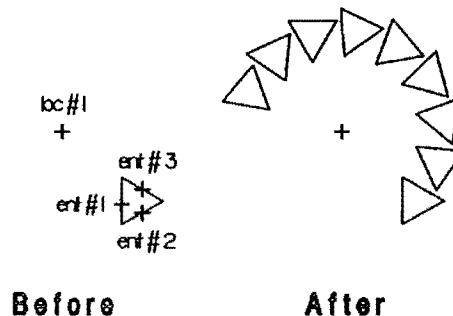


fig. (136) Rotating Entities

Transformation

Mirroring Entities



Primary

Command: TRANSFORM MIRROR

Modifiers: None

Command: TRANSFORM MIRROR: ent #1...ent #n, [;], loc #1, loc #2

Mode: 2-D and 3-D

ent #1...ent #n: One or more entities to mirror. You may mirror all entity types except 2-D text.

loc #1, loc #2: These two locations define the end points of the mirror line.

Description: This command mirrors a selected group of entities across a definable reflection line. The opposite image of the selected entities is created on the other side of the line.

Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities are mirrored across the line defined by location #1 and location #2. A mirrored image of the selected entities is created on the opposing side of the line. In 3-D mode, location #1 and location #2 define a plane that travels infinitely along the current Z axis, as defined by the active GCP.

Note: The TRANCOPY modifier in the Tools menu affects this command.

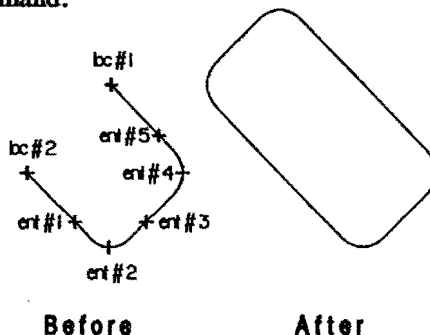


fig. (137) Mirroring Entities

Modifying Entities



Primary

Command: TRANSFORM MODIFY

Modifiers:

CHARAD



CHALOC



CHAPEN



CHATWID *



CHATHEI *



CHATROT *



CHATSLA *



CHATFONT *



CHALAYER



CHATSPACE *



EDITTEXT *



TXTORIENT *



Description: Modify changes a specified attribute of entities. Changes marked with an asterisk ("*") apply only to text entities and dimension text. You must release a dimension before it can be modified (see CREATE RELEASE, Chapter 12).

To change an entity's diameter, use the CHARAD (change radius) command.

Warning: You cannot undo a TRANSFORM MODIFY command!

Transformation

Changing Entity Radius



Command: TRANSFORM MODIFY CHARAD (rad): ent #1...ent #n

Mode: 2-D and 3-D

rad: Radius used for all selected entities.

ent #1...ent #n: One or more circles, arcs or fillets to have their radius modified.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities have their radius changed to that of "rad".

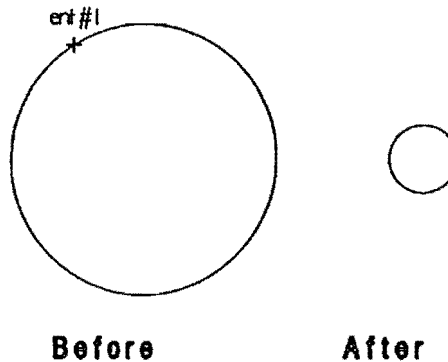
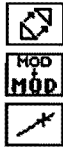


fig. (138) Entity Radius

Changing Entity Location



Command: TRANSFORM MODIFY CHALOC: ent #1...ent #n, [;]
loc #1

Mode: 2-D and 3-D

ent #1...ent #n: One or more entities to have their location modified. All entity types are permitted.

loc #1: Location which the selected entities reference. The effect this location has on the selected entity depends on the entity type. All entities are valid except b-spline and Bézier curves, and 2-D solids.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities, except lines, are moved to this new location. If the selected entity is a line, the end of the line closest to location #1 is moved to location #1.

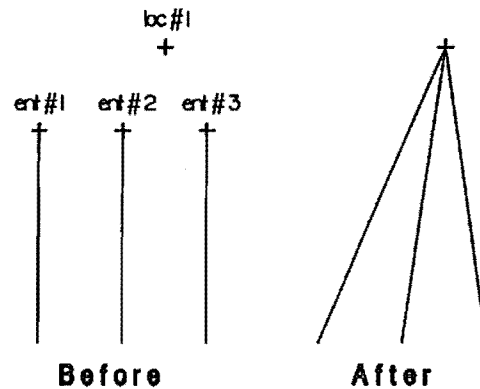
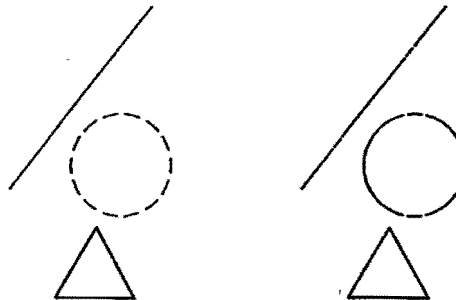


fig. (139) Entity Location

Changing Entity Pen



- Command:** TRANSFORM MODIFY CHAPEN (pen) (weight)
(line style): ent #1...ent #n
- Mode:** 2-D and 3-D
- pen:** Pen number (1 to 9, A to F), used for all selected entities.
- weight:** Line weight (1 to 3) for all selected entities.
- line style:** The line style for all selected entities. Click on the right and left arrows to scroll the style names. Defining line styles is described in Chapter 5.
- ent #1...ent #n:** One or more entities to have their pen, weight and line style modified.
- Description:** Activate the command by clicking the right mouse button, or by pressing [;].
- All selected entities have their attributes changed to that shown in the dialog box.



Before

After

fig. (140) Entity Pen

Changing Entity Layer



Command: TRANSFORM MODIFY CHALAYER (lay):
ent #1...ent #n

Mode: 2-D and 3-D

lay: Layer (0 to 255, or name), to which the selected entities are moved.

ent #1...ent #n: One or more entities to move to the selected layer. All entity types are permitted.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities have their layer changed to that of "lay". Essentially, they are moved to that layer. To copy an entity to another layer, use EDITLAYER COPYLAYER (see Chapter 15).

Changing Text Width



Command: TRANSFORM MODIFY CHATWID (wid):
ent #1...ent #n

Mode: 2-D and 3-D

wid: Width of a single text character box, used for all selected entities. DynaCADD assumes that the value is either in inches or millimeters depending on the currently selected unit type, English or Metric, respectively. The selected value is automatically readjusted to the drawing scale.

ent #1...ent #n: One or more text entities to have their character width modified.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Enter the new width value in the pop up calculator. All selected text entities have their character width changed to that of "wid".

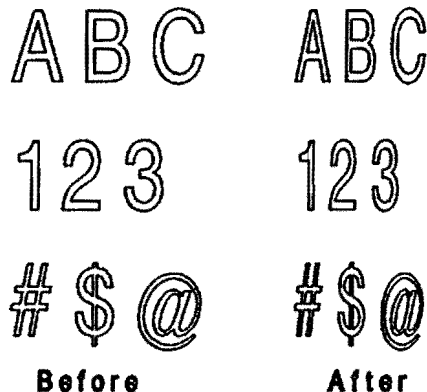


fig. (141) Text Width

Changing Text Height



Command: TRANSFORM MODIFY CHATHEI (hei):
ent #1...ent #n

Mode: 2-D and 3-D

hei: Height of a single text character box, used for all selected entities. DynaCADD assumes that the value is either in inches or millimeters depending on the currently selected unit type, English or Metric, respectively. The selected value is automatically readjusted to the drawing scale.

ent #1...ent #n: One or more text entities to have their character height modified.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Enter the new height value in the pop up calculator. All selected text entities have their character height changed to that of "hei".

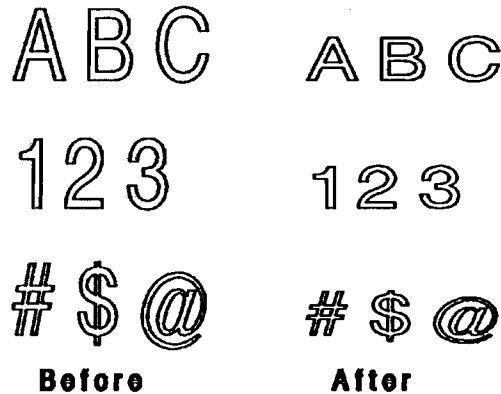
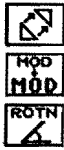


fig. (142) Text Height

Transformation

Changing Text Rotation



Command: TRANSFORM MODIFY CHATROT (rot):
ent #1...ent #n

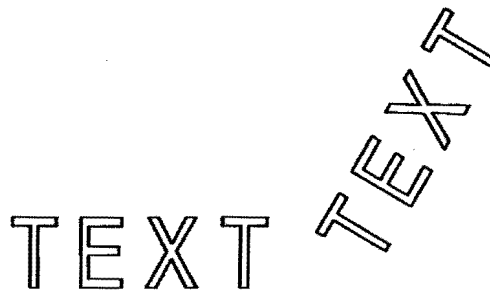
Mode: 2-D and 3-D

rot: Rotation (-360 to 360 degrees), used for all selected text entities.

ent #1...ent #n: One or more text entities to have their rotation modified.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Enter the rotation angle into the pop up calculator. All selected text entities have their rotation changed to that of "rot". In 3-D mode, the rotation is applied about the X-Y axes, defined by the coordinate plane on which the 3-D text entity lies.



Before

After

fig. (143) Text Rotation

Changing Text Slant



Command: TRANSFORM MODIFY CHATSLA (sla):
ent #1...ent #n

Mode: 2-D and 3-D

hei: Slant (-60 to 60 degrees), used for all selected text entities.

ent #1...ent #n: One or more text entities to have their slant modified.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Enter the slant in degrees, into the pop up calculator. All selected text entities have their slant changed to that of "sla".

TEXT TEXT

Before

After

fig. (144) Text Slant

Changing Text Spacing



- Command:** TRANSFORM MODIFY CHATSPACE (spc)
ent #1...ent #n
- Mode:** 2-D and 3-D
- spc:** The type of spacing (constant, proportional or kerned) used for all selected text entities. At the bottom of the dialog you can manually adjust the increment for character spacing. Click on the right and left arrows to increase and decrease the character spacing in tenths of a point (1/720 inch).
- ent #1...ent #n:** One or more text entities to have their character spacing modified.
- Description:** Activate the command by clicking the right mouse button, or by pressing [;].
- All selected text entities have their font style changed to that of the selected font style. Manually adjusted character spacing affects all text strings.
- The dialog is the same one described in Chapter 7, under INSERT TEXT SETSPACE.

T E X T T E X T

Before

After

fig. (145) Text Spacing

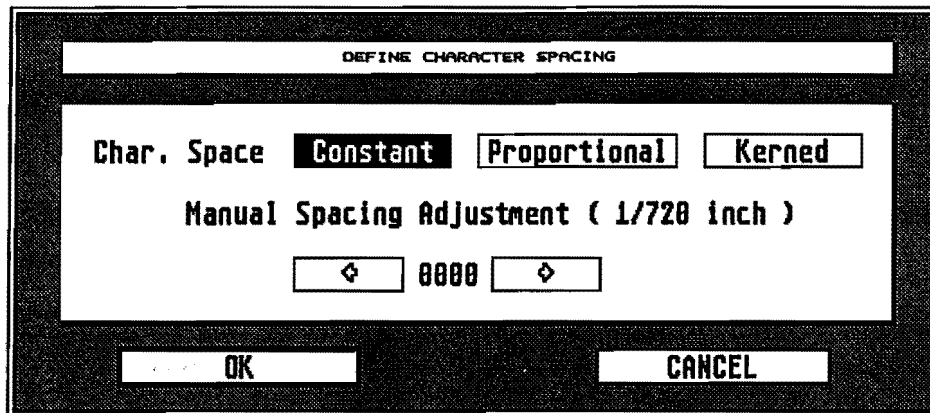


fig. (146) Character Spacing Dialog

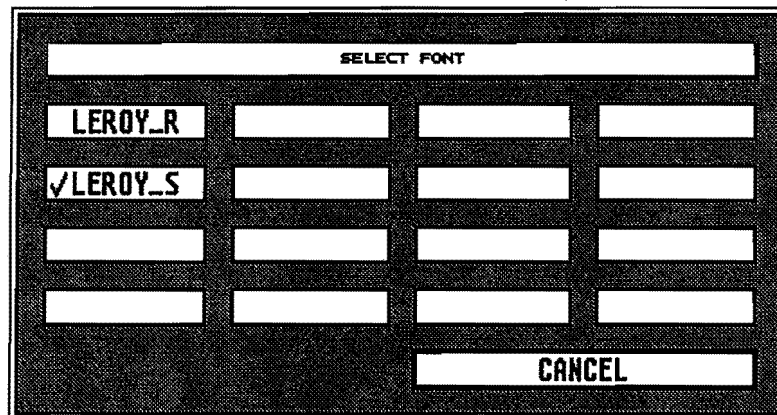


fig. (147) Font Selection Dialog

Changing Text Font



Command: TRANSFORM MODIFY CHATFONT (font)
ent #1...ent #n

Mode: 2-D and 3-D

font: After you click on the CHATFONT icon, select one of the available fonts from the dialog. The currently active font is shown with a check mark preceding its name. To select a font, click the pointer at the appropriate font name.

ent #1...ent #n: One or more text entities to have their font style modified.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

All selected text entities have their font style changed to that of the selected font style.

Triumvirate

Park Avenue

Text

Text

Before

After

fig. (148) Text Font

Transformation

Editing Text



Command: TRANSFORM MODIFY EDITTEXT ent #1

Mode: 2-D and 3-D

ent #1: The text to edit.

Description: After you select the entity, the text editor appears. Use the editing keys to change the text as required. Press [Insert] to accept the text and any changes.

See Chapter 1 for a description of the text editor, or press [Help].

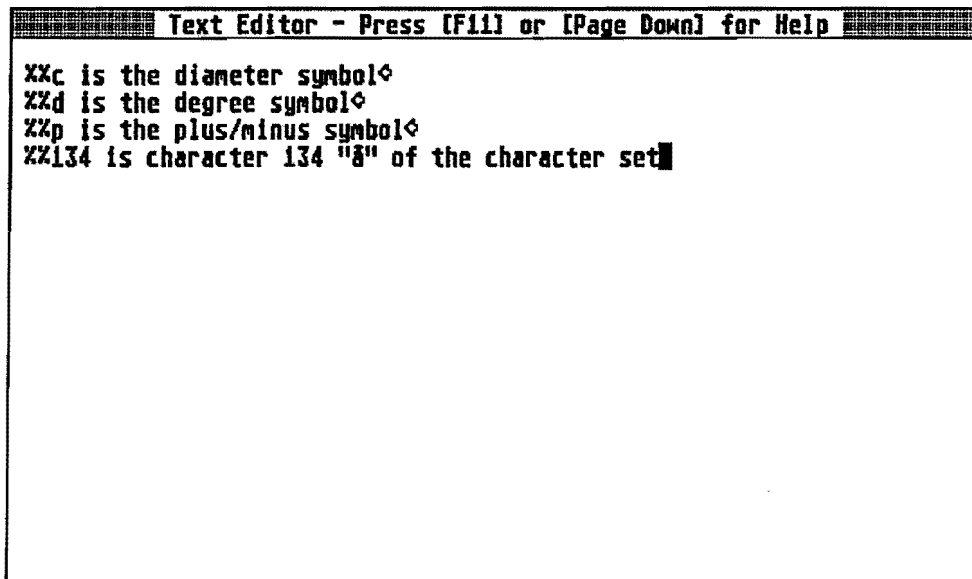


fig. (149) Text Editing

Changing Text Orientation



- Command:** TRANSFORM MODIFY TXTORIENT ent #1, loc #1, loc #2
- Mode:** 2-D and 3-D
- ent #1:** The text entity to change.
- loc #1, loc #2:** Start and end points of the new text baseline.
- Description:** Select a text entity to reorient. Then select two 2-D locations as the start and end points for the new text baseline. Click the right mouse button to change the text. The current font, slant, height, etc. are kept. The actual text size does not change size; it only changes orientation and/or moves its position.
- TXTORIENT uses the current system justification (see Insert Text, Chapter 7) and the reoriented text is given whatever is current, no matter what it had originally.
- Press [Undo] to restore the text to its original baseline.

Text

Before

Text⁺ loc #2
loc #1

After

fig. (150) Text Orientation

Transformation

Stretching Entities

**Primary****Command:** TRANSFORM STRETCH**Modifiers:** None**Command:** TRANSFORM STRETCH ent #1...ent #n, [:] loc #1, loc #2**Mode:** 2-D and 3-D**ent #1...ent #n:** One or more entities to stretch.**loc #1, loc #2:** These two locations define a stretch ratio along each axis. The first location is considered the "from" point, and the second location is the "to" point.**Description:** This command stretches a group of selected entities along any or all axes, by a definable ratio.

Activate the command by clicking the right mouse button, or by pressing [:].

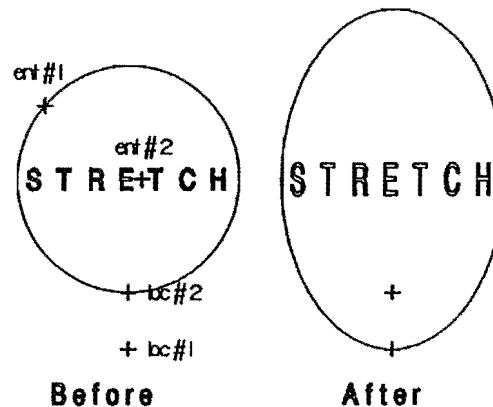


fig. (151) Stretching Entities

All selected entities have their aspect modified. The amount of stretch applied to the entities is controlled by location #1 and location #2, which define a ratio along each axis of the amount of stretch applied in each direction. Circles, Arcs and Fillets that are stretched have their base entity type changed to Ellipses and Elliptical Arcs. Although this command may prove invaluable at times, it does require some experience. We recommend that you experiment on a non-critical drawing, before using it on a part or drawing of any importance.

Note: The TRANCOPY icon affects this command.

Transformation

Scaling Entities



Primary

Command: TRANSFORM SCALE



Modifiers: None

Command: TRANSFORM SCALE ent #1...ent #n, [:] loc #1, loc #2

Mode: 2-D and 3-D

ent #1...ent #n: One or more entities to scale. You can scale all entity types, except points.

loc #1, loc #2: These two locations define a scaling ratio. The first location is the "from" point, and the second location is the "to" point.

Description: Scaling resizes selected entities by a defined ratio.

Activate the command by clicking the right mouse button, or by pressing [:].

All selected entities are resized by an ratio determined by location #1 and location #2. The center point of all selected entities is used as a reference point. The ratio is determined by the distance of location #1 and location #2 to that reference point.

Note: The TRANCOPY icon affects this command.

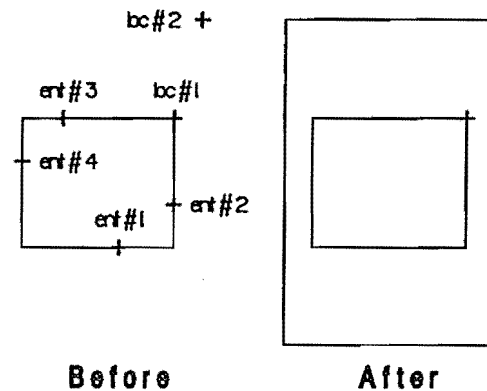


fig. (152) Scaling Entities

Creating Entity Offsets

**Primary****Command:** TRANSFORM OFFSET**Modifiers:**

THROUGH



DISTANCE



SOLIDPATH



TRACEPATH



TRACEENDS

**Description:**

Offset duplicates the entity at a specific location or distance from the original. The offset can also trace the entity on both sides.

Transformation

Offsetting Entities at a Location



Command: TRANSFORM OFFSET THROUGH ent #1, loc #1



Mode: 2-D and 3-D



ent #1: The entity to offset. Only line, arc, circle, ellipse and elliptical arc are valid entity types.

loc #1: The location of the duplicate (offset) entity. This also determines on which side of the entity the offset is placed.

Description: This creates a duplicate of the selected entity parallel to the original, on the same side and distance as that of location #1.

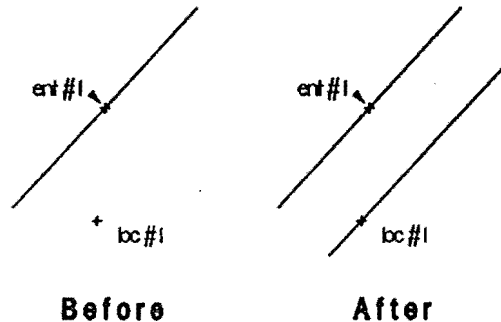


fig. (153) Transform Offset Through

Offsetting Entities at a Distance



Command: TRANSFORM OFFSET DISTANCE (dist) ent #1,
loc #1

Mode: 2-D and 3-D

dist: The distance from the original where the offset is placed.
This must be a positive number.

ent #1: The entity to offset. Only line, arc, circle, ellipse and
elliptical arc are valid entity types.

loc #1: The side at which the duplicate (offset) entity appears.

Description: This creates a duplicate of the selected entity parallel to
the original, at distance (dist), on the side of the original
indicated by location #1.

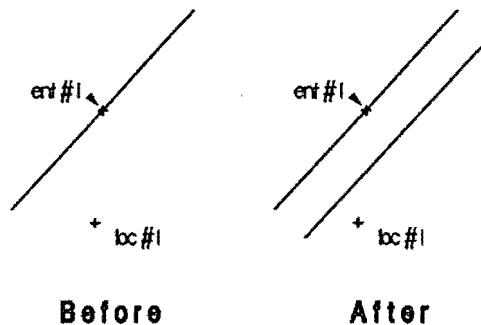


fig. (154) Transform Offset Distance

Transformation

Tracing a Solid Path Around Entities



Command: TRANSFORM OFFSET SOLIDPATH (thick)
ent #1...ent #n



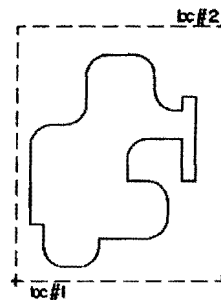
Mode: 2-D



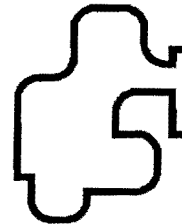
thick: The thickness of the offset path traced around the entities, measured from the original entity outward.

ent #1...ent #n: The entities to trace. If more than one entity is selected, they must all be connected in a contiguous chain.

Description: A solid path is traced around both sides of the selected entities, at thickness (thick) from and parallel to the original entities.



Before



After

fig. (155) Transform Offset Solidpath

Tracing a Path Around Entities



Command: TRANSFORM OFFSET TRACEPATH (dist)
ent #1...ent #n

Mode: 2-D

size: The distance of the offset path traced around the entities, measured from the original entity outward.

ent #1...ent #n: The entities to trace. If more than one entity is selected, they must all be connected in a contiguous chain.

Description: A path is traced around both sides of the selected entities, at distance (dist) from and parallel to the original entities.

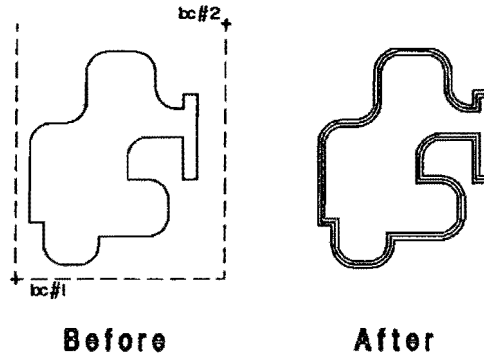


fig. (156) Transform Offset Tracepath

Transformation

Tracing a Path Around Entities and Enclose the Ends



Command: TRANSFORM OFFSET TRACEENDS (dist)
ent #1...ent #n

Mode: 2-D

size: The distance of the path traced around the entities, measured from the original entity outward.

ent #1...ent #n: The entities to trace. If more than one entity is selected, they must all be connected in a contiguous chain.

Description: A path is traced around both sides of the selected entities, at distance (dist) from and parallel to the original entities. This is the same as TRACEPATH, above, except that the ends of each entity are enclosed.

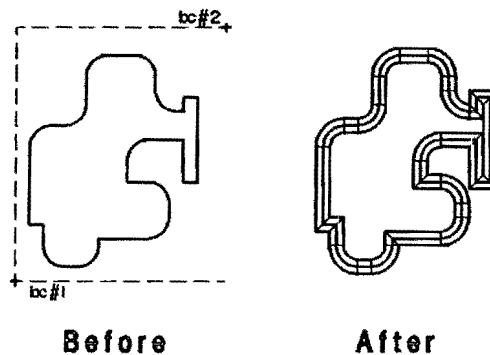


fig. (157) Transform Offset Trace Ends

Generating Points Along a Path


Primary
Command: TRANSFORM EQUIDIST

Modifiers: None

Command: TRANSFORM EQUIDIST (numpoi) ent #1...ent #n

Mode: 2-D and 3-D

numpoi: The number of points to generate along the entity path.

ent #1...ent #n: One or more lines, circles, arcs or fillets, used to define the path along which the points are placed.

Description: EQUIDIST equally spaces a specified number of points along a path as defined by a selected group of entities.

Activate the command by clicking the right mouse button, or by pressing [;].

"numpoi" points are equally spaced along a vector defined by one or more selected entities. If more than one entity is selected, each entity must share a common end point with the entity chosen previously, thereby creating an attached "chain" of entities.

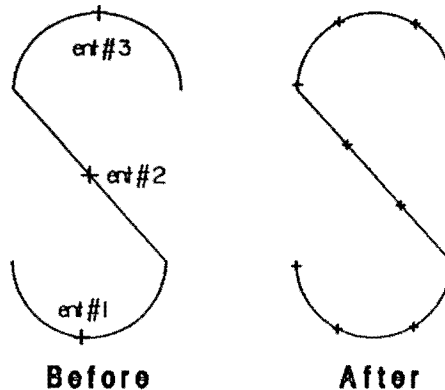


fig. (158) Transform Equidist

Projecting Entities



Primary

Command: TRANSFORM PROJECT



Modifiers: PROJDEPTH (flag) 

PROJSCALE (flag) 

PROJFACE (flag) 

Flags:

PROJDEPTH The depth to which the entities are to extrude along the Z axis. The default is 1.0.

PROJSCALE A ratio defining the constantly changing scale to apply to the entities as they are being extruded. The default is 1.0.

PROJFACE Generate 3-D faces along the extruded entities.

Description: Project extrudes a selected group of entities along the Z axis a defined depth and at a defined scale.

Projecting Entities Along the Z-Axis.



Command: TRANSFORM PROJECT (flags) ent #1...ent #n

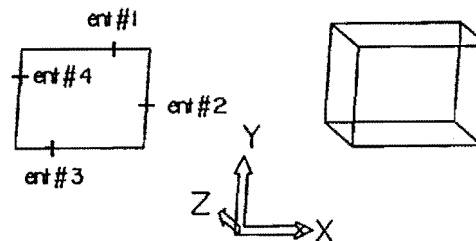
Mode: 3-D only

Flags: See primary TRANSFORM PROJECT command.

ent #1...ent #n: One or more entities selected to project. All 3-D entity types are allowed.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities are extruded along the current Z axis (defined by the active GCP) to a depth of PROJDEPTH. The entities also have a constant scaling factor applied, so that they are PROJSCALE times their original size, upon completion of the extrusion. Projection lines are generated at each entity end point.



Before

After

fig. (159) Project Entities along a Z Axis

Transformation

Revolving Entities



Primary



Command: TRANSFORM REVOLVE

Modifiers:

REVANG

(flag)



REVNCOPY

(flag)



REVN_MESH

(flag)



REVFACE

(flag)



Flags:

REVANG

The total angle (-360 to +360 degrees), that the entities are revolved.

REVNCOPY

The number of additional profiles the entities create as they are being revolved. This value must be greater than zero.

REVN_MESH

The number of supporting arcs that are placed perpendicular to the revolve vector. A value of one places mesh at every end point of each selected entity.

REVFACE

Generate 3-D faces along the revolved entities.

Description:

REVOLVE rotates selected entities about a definable revolution vector. You can also specify an optional supporting mesh (a series of circular rings created around the revolved entities, perpendicular to the axis of revolution, figure displayed below).

Revolving Entities about a Vector



Command: TRANSFORM REVOLVE (flags) ent #1...ent #n, [:]
loc #1, loc #2

Mode: 3-D only

Flags: See primary TRANSFORM REVOLVE command.

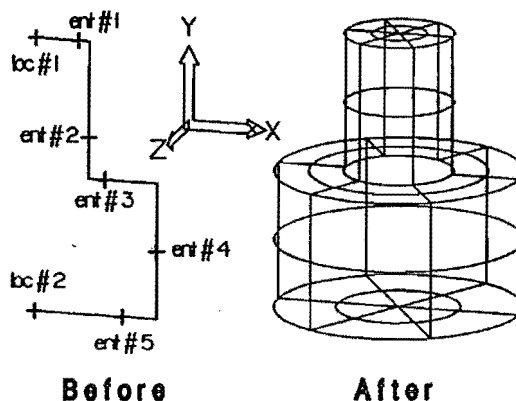
ent #1...ent #n: One or more entities to revolve.

loc #1, loc #2: These two locations define a 3-D vector, which can be thought of as a pole that the selected entities spin about.

Description: Activate the command by clicking the right mouse button, or by pressing [:].

All selected entities revolve about the vector defined by location #1 and location #2, a total of REVANG degrees. Duplicate profiles of the selected entities are generated REVNCOPY times and are equally spaced between the total angle. You can generate an optional supporting mesh, placed perpendicular to the revolution vector. Each entity has REVNMESH number of arcs placed along its length. If a value of one is selected, the supporting mesh is placed at each entity end point.

fig. (160)
Revolving Entities



Sweeping Entities

**Primary****Command:** TRANSFORM SWEEP**Modifiers:** SWEEPFACE (flag)**Command:** TRANSFORM SWEEP ncopy (flag) ent #1...ent #n [;]**Mode:** 3-D only**Flag:** SWEEPFACE Generate 3-D faces along the projected sweep path.**ncopy:** The number of entity profiles to generate on each entity in the path. If a value of one is selected, profiles are generated at the end of each entity in the path.**ent #1...ent #n:** One or more 3-D entities to sweep along a path. Ent #2 and up define the path the previously selected entities follow. If more than one entity is selected, each entity must share a common end point with the entity previously chosen, thereby creating an attached chain of entities. All entity types, except text and points, are permitted to form the chain.**Description:** Sweep generates entities along the Z axis, at a scale defined by a selected group of entities.

Activate the command by clicking the right mouse button, or by pressing [;].

A selected group of entities are projected along a path defined by a secondary group of entities. The scale and direction is defined by the path as it travels along the Z axis. The initial scaling factor is set at one, at the end

point of the first entity in the path. This scaling factor varies as the path deviates from the calculated center of the initial group of selected entities. NCOPY number of profiles are generated by the command. If NCOPY is equal to one, profiles of the selected entities are generated at the end points of every entity in the path. You can also generate the 3-D faces along the sweep surface by activating SWEEPFACE.

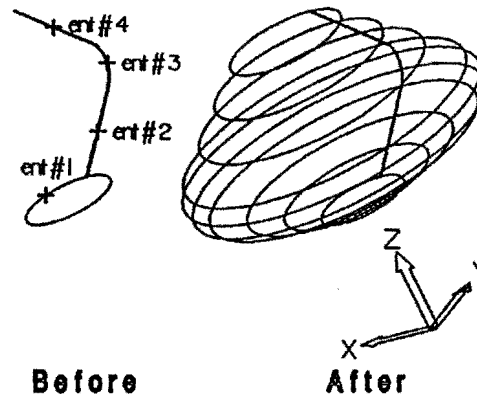


fig. (161) Sweeping Entities

Linear Arrays of Entities



Primary

Command: TRANSFORM ARRAY

Modifiers:

INCX (flag)	
NX (flag)	
INCY (flag)	
NY (flag)	
INCZ (flag)	
NZ (flag)	

Flags:

INCX	The distance to increment along the X axis for each repetition of the command. The default is 1.0.
NX	The total number of columns required along the X axis. The default is 1.0 unit.
INCY	The distance to increment along the Y axis for each repetition of the command. The default is 1.0.
NY	The total number of columns required along the Y axis. The default is 1.0 unit.
INCZ	The distance to increment along the Z axis for each repetition of the command. The default is 1.0.
NZ	The total number of columns required along the Z axis. The default is 1.0 unit.

Note: The INCZ and NZ flags are only applicable in 3-D mode.

Description: ARRAY moves and copies a group of entities from one location to another, along each of the axes a definable number of times.

An array is a group of entities copied along an axis. A circular array is a group of entities copied in a circular pattern.

Creating Arrays of Entities



Command: TRANSFORM ARRAY (flags) ent #1...ent #n

Mode: 2-D and 3-D

Flags: See primary TRANSFORM ARRAY command.

ent #1...ent #n: One or more entities for creating the linear array. You can select all entity types.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The array entity command creates a one, two, or three dimensional linear array of the selected entities.

The entities are incremented along the X, Y and Z axes, by a distance defined by the INCX, INCY and INCZ flags. The total number of columns along each of the axes, is defined by the NX, NY and NZ flags. A value greater than one for the NX, NY or NZ flags is required to create new entities, since the selected entities themselves are considered as the first column.

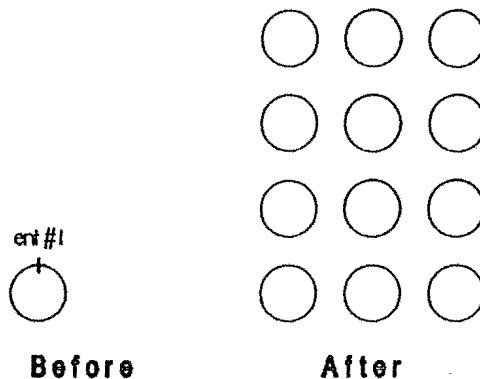


fig. (162) Creating Arrays

Masking Entities

**Primary****Command:** TRANSFORM MASK**Modifiers:** None**Command:** TRANSFORM MASK ent #1...ent #n**Mode:** 2-D and 3-D**ent #1...ent #n:** One or more entities to mask. All entity types are allowed.

Description: MASK temporarily hides a selected group of entities. By hiding a portion of the drawing that is currently not being accessed, the drawing is simplified. This technique also has the added benefit of speeding up the repaint and access time, since fewer entities require updating.

Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities are hidden. The entities are unavailable for reference until restored using the UNMASK command.

Unmasking Entities

**Primary****Command:** TRANSFORM UNMASK**Modifiers:** None**Command:** TRANSFORM UNMASK**Mode:** 2-D and 3-D

Description: UNMASK is the counterpart of the MASK command. It restores all previously masked entities in order that you can select them again, for reference.

In 2-D mode, only 2-D entities are restored, and in 3-D mode, only 3-D entities are restored.

Transforming 3-D Entities to 2-D Entities

**Primary****Command:** TRANSFORM BURNVIEW**Modifiers:** None**Command:** TRANSFORM BURNVIEW view, ent #1...ent #n**Mode:** 3-D only**view:** Selected view which defines the 2-D representation of the 3-D entities.**ent #1...ent #n:** One or more entities to transform to 2-D. You can select all entity types, except text.**Description:** BURNVIEW transforms a selected group of 3-D entities into a corresponding group of 2-D entities.

Activate the command by clicking the right mouse button, or by pressing [;].

The 2-D representation of all selected 3-D entities is created. The selected view's rotation and scale are used to project the 3-D entities onto the drawing. The selected 3-D entities remain unmodified. The transformation actually creates a 2-D image of the selected entities.

Be careful when selecting entities for BURNVIEW. If you choose entities BYALL, you may be overlapping entities as entities hidden in behind those you can see are also converted. Select entities by clicking to be sure you get what you want.

Adjusting Alignment of Entities



Primary

Command: TRANSFORM ALIGN



Command: TRANSFORM ALIGN ent #1..ent #n, [;]
loc #1, loc #2, loc #3, loc #4

Mode: 2-D and 3-D

ent #1..ent #n: Entities to be aligned. This is equivalent to a TRANSFORM TRANSLATE and ROTATE Command.

loc #1: Location #1 defines the initial translation point.

loc #1, loc #2: The deviation between location #2 and location #1 defines the baseline of the rotation to be performed.

loc #1, loc #3: The selected entities will be translated along a vector defined by the deviation between location #3 and location #1.

loc #3, loc #4: The deviation between location #4 and location #3 defines the angular deviation at which the selected entities will be rotated.

Description: The selected entities will be moved or copied along a vector from location #1 to location #3.

The selected entities are then rotated about location #3, an angular deviation equal to the angle between the vectors (location #1 - location #2) and (location #3 - location #4).

View Manipulation ... Chapter 11

As we discussed previously, 3-D entities actually exist in the space behind the drawing page. If no views are active, you are unable to see the full 3-D entities. Views provide an X-ray type ability to look beyond the surface of the drawing.

Keep in mind that all views look in on the same part and that an entity transformed in one view is changed in all others simultaneously.

DynaCADD supports up to four active, simultaneous views. The more views that are active, the longer it takes to regenerate the screen. It's a good idea to only use one or two views in a design session, then, if necessary, create the extra views for output.

Views are necessary in 3-D. You cannot view 3-D entities without at least one view open. When you change to 2-D mode, the views remain but the GCP indicator disappears. While the commands below work in 2-D mode, most are really only relevant to 3-D, since views do not affect 2-D entities that co exist on the drawing. References in this chapter to 2-D locations are to the 2-D plane, not 2-D mode.

All commands related to view manipulation are combined under the EDITVIEW icon. There are also several GCP related commands under the CREATE icon.

Deleting a View

**Primary****Command:** EDITVIEW DELVIEW**Modifiers:** None**Command:** EDITVIEW DELVIEW view #1**Mode:** 2-D and 3-D**view #1:** The active view to remove from the drawing.**Description:** DELVIEW removes a selected view from the drawing. This has no effect on the entities.

The selected view is removed (deleted) from the drawing. Any entities visible at the rotation defined by the selected view, are no longer displayed.

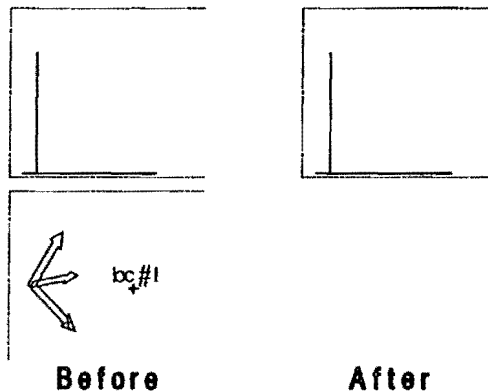


fig. (163) Deleting a View

Creating a View

**Primary****Command:** EDITVIEW DEFVIEW**Modifiers:**

CVIEWGCP (flag)



CVIEWWSCL (flag)

**Flags:**

CVIEWGCP Defines the three axis primary rotation. Select a 3-D rotation, by clicking the pointer on the required GCP name. The default GCP for the DEFVIEW command is Top.

CVIEWWSCL Defines the view's scale. This ratio is applied to the view after the drawing scale is applied. If a scale of 0.5 is selected, the view shows the part at one half the original size, and a scale of two shows the part at twice the original size. The default scale is 1.0.

Description:

DEFVIEW creates a new view at a specified scale and three axis rotation. Use this command to automatically create orthographic views.

Defining a View

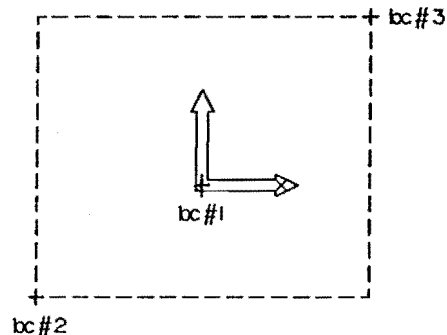


- Command:** EDITVIEW DEFVIEW (flags) loc #1, loc #2, loc #3
- Mode:** 2-D and 3-D
- Flags:** See primary EDITVIEW DEFVIEW command.
- loc #1:** This is the page origin of the view, a 2-D location on the drawing where the origin of the 3-D part is placed.
- loc #2, loc #3:** These locations define a rectangular viewing clip, graphically displayed as a dotted rectangle. If View Clipping in the Tools menu is On, the entities in the view are clipped inside this frame. Click the right mouse button or press [;] after you select location #1 to automatically define the view clip as the full size of the drawing page.

Description: A view is created and all 3-D entities are displayed at the rotation defined by CVIEWGCP. The origin of the view is placed at location #1. The viewing clip is a rectangle defined by location #2 and location #3. All 3-D entities are clipped to this view, providing the View Clipping option in the Tools menu is On. The scale of the new view is defined by CVIEWWSCL.

You can change the view's rotation and scale by using additional commands under the EDITVIEW icon (see below); you do not need to create new views for this purpose.

fig. (164)
Defining a View



Changing a View to a GCP Rotation

**Primary****Command:** EDITVIEW VIEWGCP**Modifiers:** None**Command:** EDITVIEW VIEWGCP gcp view #1**Mode:** 2-D and 3-D**gcp:** Use the pointer to choose the required GCP from the Select GCP dialog box. All available GCPs are listed.**view #1:** The active view to have its rotation changed to that of the selected GCP.**Description:** VIEWGCP rotates a view's rotation to that defined by a selected GCP.

The selected view is redisplayed, with its rotation changed to that of "gcp". This command makes it possible to change a top view to a front or side view without creating another view.

SELECT GEOMETRIC CO-ORDINATE PLANE			
☒ Top	Rear		
Front	Left		
Right	Iso		
Bottom			
CANCEL			

fig. (165) Selecting a GCP

Changing a View to Absolute Rotation

**Primary****Command:** EDITVIEW ROTVIEW**Modifiers:** AXROT (flag)

AYROT (flag)



AZROT (flag)

**Flags:****AXROT** The absolute rotation (-360 to 360 degrees) about the X axis. The default is 0 degrees.**AYROT** The absolute rotation (-360 to 360 degrees) about the Y axis. The default is 0 degrees.**AZROT** The absolute rotation (-360 to 360 degrees) about the Z axis. The default is 0 degrees.**Description:**

ROTVIEW rotates a view's rotation to that defined by the available flags. These specify an absolute rotation about each of the three axes.

Rotating a View



- Command:** EDITVIEW ROTVIEW (flags) view #1
- Mode:** 2-D and 3-D
- Flags:** See primary EDITVIEW ROTVIEW command.
- view #1:** The active view has its rotation changed to that defined by the AXROT, AYROT and AZROT flags.
- Description:** The selected view is redisplayed with its rotation changed to that defined by AXROT, AYROT and AZROT. The default rotation for all flags is zero degrees.

This command rotates the view, not the entities displayed in that view, then snaps the view perpendicular to the screen after. Because of this view snap, effects appear reverse. For example, applying a rotation of 45 degrees about the Z axis makes the entities appear to have been rotated -45 degrees, because you rotate the view, not the actual entities.

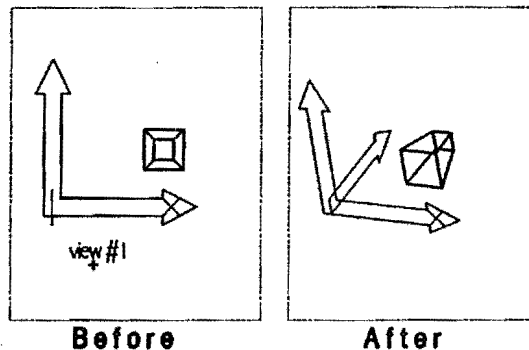


fig. (166) Rotate a View

Revising a View Clip

**Primary****Command:** EDITVIEW REVCLIP**Modifiers:** None**Command:** EDITVIEW REVCLIP view #1, loc #1, loc #2**Mode:** 2-D and 3-D**view #1:** The active view to modify.**loc #1, loc #2:** Two diagonally opposing 2-D locations which define a window to replace the current view clipping area.**Description:** REVCLIP modifies a viewing clip.

The selected view is redisplayed with its viewing clip changed to the window defined by location #1 and location #2. Providing the View Clipping option under the Tools menu is on, the entities currently displayed in the selected view are redisplayed, contained within the new clipping area.

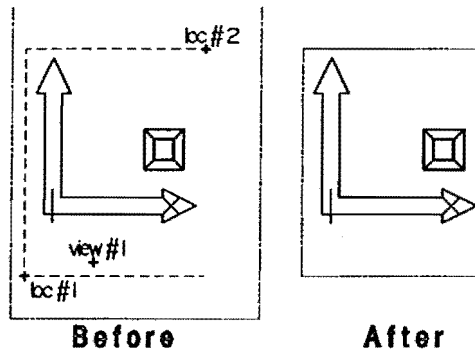
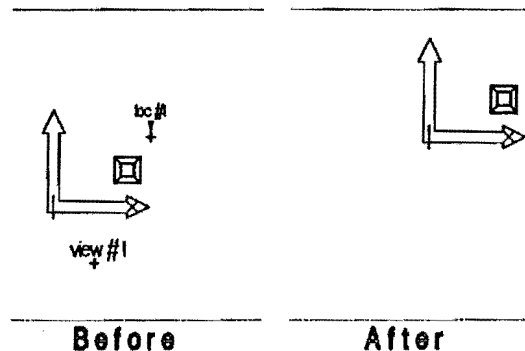


fig. (167) Revising a View Clip

Revising a View Origin

**Primary****Command:** EDITVIEW REVORG**Modifiers:** None**Command:** EDITVIEW REVORG view #1, loc #1,**Mode:** 2-D and 3-D**view #1:** The active view to modify.**loc #1:** A 2-D location where the new page origin of the view is placed.**Description:** REVORG modifies the page origin of a view.

The selected view is redisplayed, with its origin at location #1. The origin of the view is the location the GCP uses as its X0, Y0, Z0 point in that particular view.

*fig. (168) Revising a View Origin*

Revising a View Scale

**Primary****Command:** EDITVIEW REVSCALE**Modifiers:** None**Command:** EDITVIEW REVSCALE scl view #1**Mode:** 2-D and 3-D**scl:** Enter the viewing scale into the pop up calculator when you click REVSCALE.**view #1:** The active view to modify.**Description:** REVSCALE modifies the scale of a view.

The selected view is re-displayed, with its scale set to "scl".

Do not confuse the view scale with that of the drawing scale. The drawing scale applies to all 2-D and 3-D entities and all locations selected during the design session. A view's scale is used to modify only the 3-D entities being displayed in that particular view. The actual entities are not modified or scaled in any way.

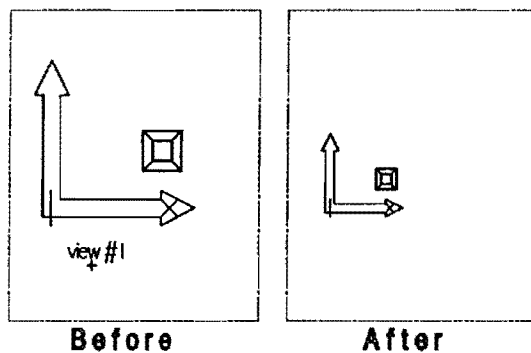


fig. (169) Revising a View Scale

Resetting a View



Primary

Command: EDITVIEW RESETVIEW

Modifiers: None

Command: EDITVIEW RESETVIEW view #1

Mode: 2-D and 3-D

view #1: The active view to reset.

Description: RESETVIEW resets a view's rotation, zoom factor, scale and scroll factor to the initial definitions used when creating the view.

The selected view is redisplayed at its initial rotation and scale. The view's zoom factor is set to 1.0, scroll is reduced and the original view settings restored. The view is displayed using the same parameters as at the time it was created.

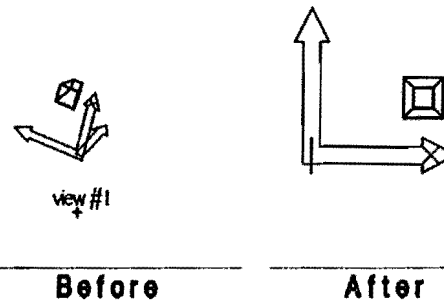


fig. (170) Reset View to Original Settings

Dynamic Rotation of a View



Primary

Command: EDITVIEW DYNAMIC

Modifiers: None

Command: EDITVIEW DYNAMIC view #1

Mode: 2-D and 3-D

view #1: The view to change.

Description: Dynamic view rotates, zooms and scrolls a view in real time, as you click the increment arrows in the dynamic calculator.

Use the dynamic calculator to interactively change the rotation, zoom and scroll factor or the view. All rotations use the origin of the part as the center of the rotation.

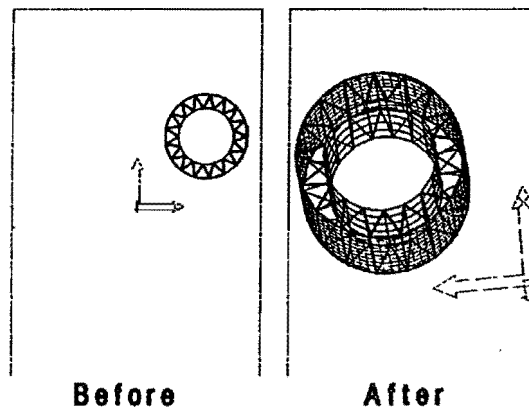


fig. (171) Dynamically change a View

Using the Dynamic Calculator

The center RESET button in each of the three areas cancels all actions in that area, and resets the view to the state active when the dynamic calculator appeared. The small arrows at the top of each box change the factor by which the view is changed. Click on the triple arrow buttons to execute the change in that direction.

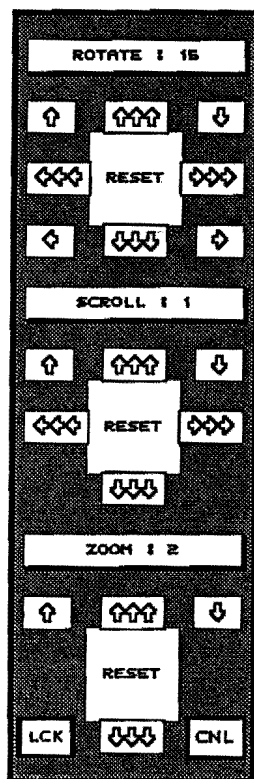


fig. (172) Dynamic Calculator

Rotation: Use the upper left and upper right buttons to raise and lower the angular rotation in single degree increments. The four outside buttons rotate the view on the X or the Y axes, by the angle shown in the center, in the direction of the individual arrows.

The left and right arrows at the bottom are for clockwise and counter clockwise rotation about the Z axis, respectively.

Scrolling: Use the upper left and upper right buttons to raise and lower the view scroll factor. A scrolling factor of one is approximately two inches on the display. Use the four outside buttons to scroll the view in the various directions by the factor displayed in the center button. The view is scrolled in the direction indicated by the four arrows.

Zooming: Use the upper left and upper right buttons to raise and lower the zoom factor of the view. The current zoom factor is multiplied by this value. Use the top center button to zoom the view in by the factor displayed in the center

View Manipulation

button. Use the lower center button to zoom the view out by the factor displayed in the center button.

Cancel: Click CNL to cancel all changes to the view and exit the Dynamic calculator.

Lock: Click LCK to lock all changes to the view and exit the Dynamic calculator. Depending on the number and type of entities in the view, this may take some time to complete.

Zooming In on an Active View


Primary
Command: EDITVIEW VIEWZIN

Modifiers: None

Command: EDITVIEW VIEWZIN view #1

Mode: 2-D and 3-D

view #1: The view to zoom closer.

Description: VIEWZIN zooms in (down) on a selected view by a factor of two. The origin of the view is used as the center of the zoom.

Do not confuse the view zoom commands with the drawing zoom commands. The drawing zoom commands bring the actual page closer. The view zoom commands only modify the appearance of the 3-D entities being displayed in the view. The selected view is zoomed closer, by a factor of two. The origin of the part is used as the center of the zoom.

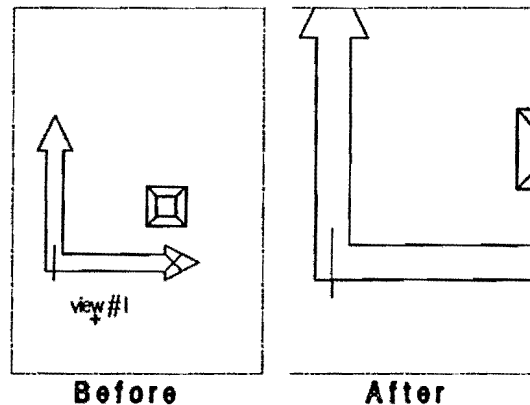


fig. (173) Zooming in on a View

Zooming Out from an Active View

**Primary****Command:** EDITVIEW VIEWZOUT**Modifiers:** None**Command:** EDITVIEW VIEWZOUT view #1**Mode:** 2-D and 3-D**view #1:** The view to zoom out.**Description:** VIEWZOUT zooms out (up) from a selected view, by a factor of two. The origin of the view is used as the center of the zoom.

Do not confuse the view zoom commands with the drawing zoom commands. The drawing zoom commands bring the actual page closer. The view zoom commands only modify the appearance of the 3-D entities being displayed in the view.

The selected view is zoomed out from, by a factor of two. The origin of the part is used as the center of the zoom.

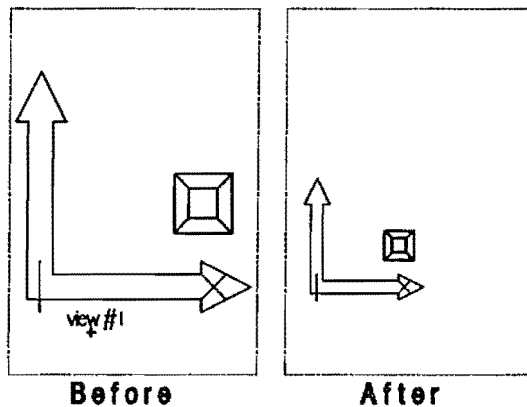


fig. (174) Zooming out on a View

Zooming an Active View to Database Extent


Primary
Command: EDITVIEW VIEWZEXT

Modifiers: None

Command: EDITVIEW VIEWZEXT view #1

Mode: 2-D and 3-D

view #1: The view to zoom to the database extent.

Description: VIEWZEXT zooms the extent of the 3-D entities to fit into a selected view. The extent of the database is an imaginary cube that encloses all entities within it. The size of the cube depends on position of the most distant entity.

An imaginary cube is constructed around the space occupied by all 3-D entities. This imaginary box is then be zoomed in or out by an amount required to fit all entities into the selected view.

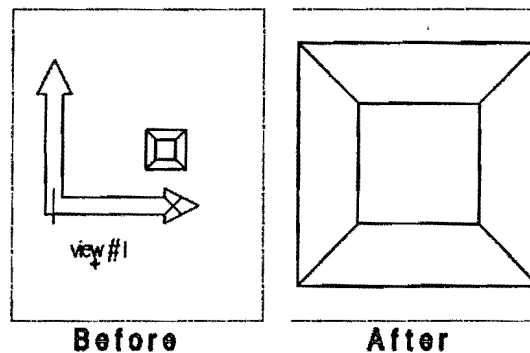


fig. (175) Zooming to Database Extent

Scrolling an Active View

**Primary****Command:** EDITVIEW SCRVIEW**Modifiers:** None**Command:** EDITVIEW SCRVIEW view #1, loc #1, loc #2**Mode:** 2-D and 3-D**view #1:** The view to scroll.**loc #1, loc #2:** Two 2-D locations which define the vector along which the view is scrolled.**Description:** SCRVIEW scrolls, or pans, an active view from one location to another.

The selected view is scrolled from location #1 to location #2, a 2-D scroll along the plane of the screen. The entities are not moved. SCRVIEW acts only on the position that the view assumes is showing the 3-D part. The 3-D part itself does not change, only the view changes.

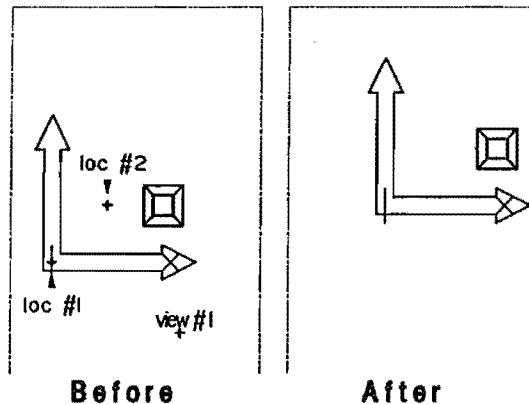


fig. (176) Scrolling a View

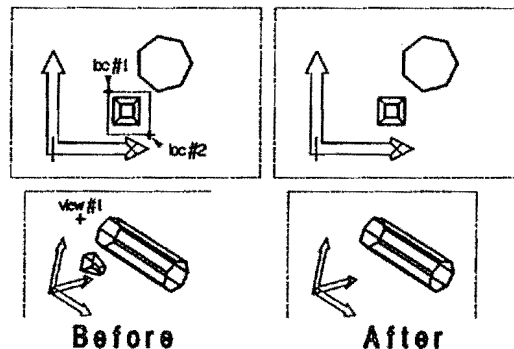
Hiding Entities in an Active View

**Primary****Command:** EDITVIEW HIDE**Modifiers:** None**Command:** EDITVIEW HIDE view #1, ent #1...ent #n**Mode:** 3-D only**view #1:** The view in which to remove the entities.**ent #1...ent #n:** One or more entities to hide in the selected view. All entities types are permitted.**Description:** HIDE temporarily removes selected 3-D entities from a view. They are displayed in all other active views.

Activate the command by clicking the right mouse button, or by pressing [;].

The entity is not deleted or modified in any way and is only affected in the selected view. It is, however, possible to select the entities in any of the other active views. Use this command to remove entities that are usually hidden in a particular view but are still required to define the part in one or more of the other orthographic views.

*fig. (177) Hide
Entities from a View*



Redisplaying Hidden Entities in an Active View



Primary

Command: EDITVIEW SEEK

Modifiers: None

Command: EDITVIEW SEEK view #1, ent #1...ent #n

Mode: 3-D only

view #1: The view in which to restore the entities.

ent #1...ent #n: One or more entities to restore in the selected view.

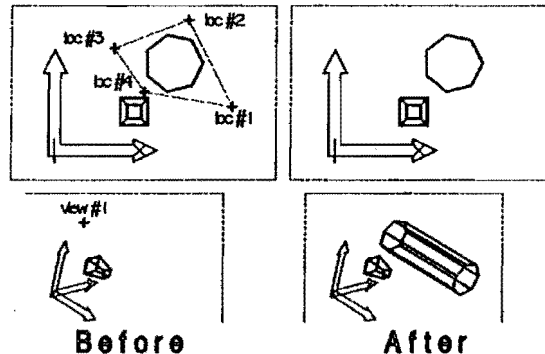
Description: SEEK restores previously hidden entities in a selected view. The entities must be visible in at least one view.

If you only have one view active, then you cannot select hidden entities (they are not visible to select!). To fix this, open another view and select the entities there.

Activate the command by clicking the right mouse button, or by pressing [;].

This is the counterpart of the HIDE command (above). After you select the view in which to view the entities, you must select the hidden entities in a view in which they are still currently active. Once the command is activated, all selected entities are redisplayed in the selected view.

*fig. (178)
Restore
Hidden
Entities in a
View*



Tool Creation and Selection ... Chapter 12

In DynaCADD you have the ability to construct several tools to suit your individual needs. This chapter deals with the construction and selection of tools, fonts and figures, including commands to create macros, images and to move the GCP. Other GCP commands are found under the EDITVIEW icon (see Chapter 11).

All creation and related selection commands are combined under the CREATE icon.

Loading Font Definitions



Primary

Command: CREATE LOADFONT



Modifiers: None

Command: CREATE LOADFONT font

Mode: 2-D & 3-D

font: Use the pointer to select the required font from the File Selector dialog.

Description: This command loads a font file from the disk.

After you select the desired font, DynaCADD loads the font definition file. The font is then listed in the Select Font dialog.

Selecting the Current Font



Primary

Command: CREATE SELFONT

Modifiers: None

Command: CREATE SELFONT font

Mode: 2-D & 3-D

font: The Select Font dialog displays all fonts that are installed in memory. Use the pointer to select a new current font.

Description: Use SELFONT to select an installed font from the dialog as the current font. All installed fonts are displayed, with the currently active font indicated by a check mark. Any text entities or dimensions you insert after this choice appear in the active font. Text previously defined in another font remains in that font.

The selected font becomes the current font. All new text entities created are drawn using the selected font definition. The current font is always displayed preceded by a check mark in the Select Font dialog.

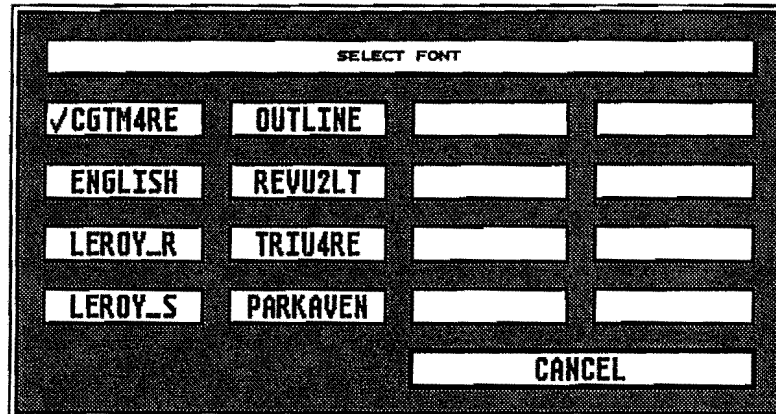


fig. (179) Font Selection

Grouping Entities Into Subfigures



Primary

Command: CREATE GROUP

Modifiers: None

Command: CREATE GROUP ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: One or more selected entities, grouped into a Subfigure. All entity types, including existing subfigures, are allowed.

Description: GROUP dynamically creates a Subfigure from all selected entities. The selected entities are then treated as a single complex entity. To break a group, use RELEASE.

Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities are grouped into a subfigure and treated as if they were all a single complex entity. When any single entity in the group is selected, all others are automatically selected.

Releasing Subfigures



Primary

Command: CREATE RELEASE

Modifiers: None

Command: CREATE RELEASE ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: One or more selected subfigures to release.

Description: RELEASE breaks all selected subfigures (see GROUP) into their individual entity types.

Activate the command by clicking the right mouse button, or by pressing [;].

This is the counterpart of the GROUP command. All selected Subfigures are broken down into their original singular entity types. Whether the Subfigure was created using the GROUP command, or inserted from a disk file, has no bearing on the RELEASE command. They are both released.

Although the GROUP command allows the selection of smaller Subfigures to create a single larger Subfigure, releasing the larger Subfigure still breaks the entire structure into its individual parts. The smaller groups are not restored.

Removing a GCP Definition



Primary Command: CREATE REMGCP

Modifiers: None

Command: CREATE REMGCP GCP

Mode: 2-D & 3-D

GCP: After you select the REMGCP icon, use the pointer to select the GCP to remove from the Select GCP dialog.

Description: REMGCP removes a GCP definition from memory.

The selected GCP is removed from memory and is no longer displayed in the GCP menu. You are not able to remove the current GCP, or any of the primary GCPs. The primary GCPs are system defined and are resident at all times. These include:

Top	Rear
Front	Left
Right	Iso (Isometric)
Bottom	

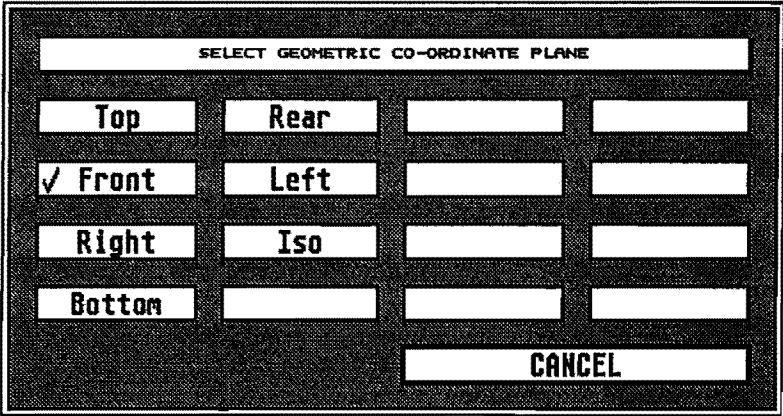


fig. (180) Selecting a GCP

Creating a GCP Definition



Primary

Command: CREATE DEFGCP

Modifiers:

GCPPLANE



GCPNORMAL



GCPVIEW



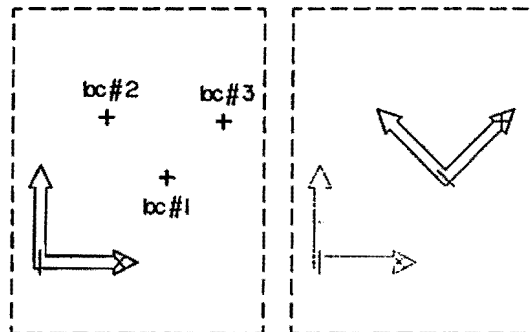
GCPARC



Description:

DEFGCP creates a Geometric Coordinate Plane. If you select the defined GCP as the current GCP, it is possible to create not only custom coordinate systems, but to define any plane in 3-D space.

New GCP definitions are installed in the GCP menu for later selection. In order to see the view with the defined GCP, either select the GCP from the menu or use EDITVIEW VIEWGCP.



Before

After

fig. (181) GCP Plane

Defining a GCP Using a Three Point Plane



Command: CREATE DEF GCP (name) GCPPLANE loc #1, loc #2, loc #3

Mode: 3-D only

name: After you select the GCP plane icon, a small dialog prompts for an eight character name for the GCP you create. If you click on the Cancel button, or fail to enter a name with one or more characters, DynaCADD cancels the command.

loc #1: The location defining the origin (X0, Y0, Z0) of the GCP you create. The following two locations (loc #2 and loc #3) use loc #1 as a reference.

loc #2: A location lying on the X axis of the GCP you are defining.

loc #3: A location lying on the Y axis of the GCP you are defining.

Description: A GCP is created and stored under the label "name". The three locations (figure on preceding page) define the origin, X axis, Y axis and plane of the new GCP. Location #1 is defined as the origin and the GCP moves to the origin defined by this location. The newly defined GCP is displayed with its origin coincident with the Part origin.

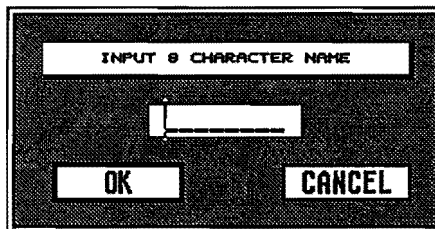


fig. (182) GCP Name Input

Defining GCP Normal to Plane



Command: CREATE DEFGCP (name) GCPNORMAL loc #1,
loc #2

Mode: 3-D only

name: After you select the GCP normal icon, a small dialog prompts for an eight character name for the GCP you created. If you click on the Cancel button, or fail to enter a name with one or more characters, DynaCADD cancels the command.

loc #1, loc #2: These locations define the vector along the Z axis of the GCP you are creating.

Description: A GCP is created and stored under the label "name". Location #1 and location #2 define the new GCP's Z axis. The GCP normal command creates a plane that lies normal (perpendicular) to the vector defined by the two locations. This command is normally used to create a plane whenever the orientation of the X and Y axes directions are of no importance.

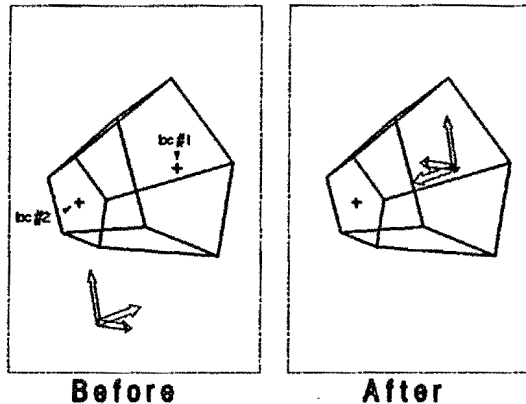


fig. (183) Normal GCP

Defining a GCP Using a View Definition



Command: CREATE DEFGCP (name) GCPVIEW view #1



Mode: 3-D only



name: After you select the GCP plane icon, a small dialog prompts for an eight character name for the GCP you create. If you click on the Cancel button, or fail to enter a name with one or more characters, DynaCADD cancels the command.

view #1: The view used as reference to define the 3-D rotation of the GCP you are creating.

Description: A GCP is created and stored under the label "name". The new GCP uses the current rotation of the selected view to define the X and Y axes. The plane is parallel to the current view. Once selected, the defined GCP is aligned with the view definition. After you dynamically rotate a view, it is sometimes useful to store the current rotation of that same view. By defining a GCP to match the current rotation of a view, you can rotate any view back to that same state by using the EDITVIEW VIEWGCP command.

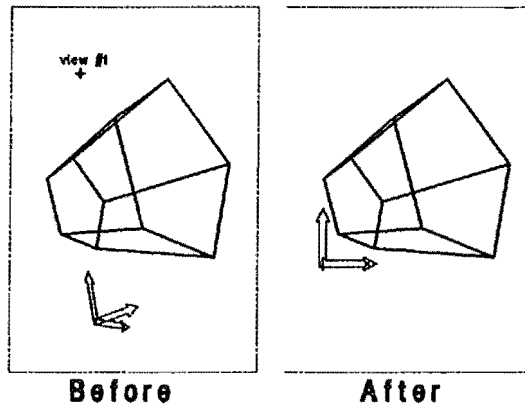


fig. (184) GCP Definition using a View

Defining a GCP Using an Arc Definition



- Command:** CREATE DEFGCP (name) GCPARC ent #1
- Mode:** 3-D only
- name:** After you select the GCP plane icon, a small dialog appears prompts for an eight character name for the GCP you create. If you click on the Cancel button, or fail to enter a name with one or more characters, DynaCADD cancels the command.
- ent #1:** Circle, arc, fillet, ellipse or elliptical arc, used as reference to define the 3-D rotation of the GCP you are creating.
- Description:** A GCP is created and stored under the label "name". The new GCP uses the 3-D rotation of the selected entity to define the X and Y axes in addition to the coordinate plane.

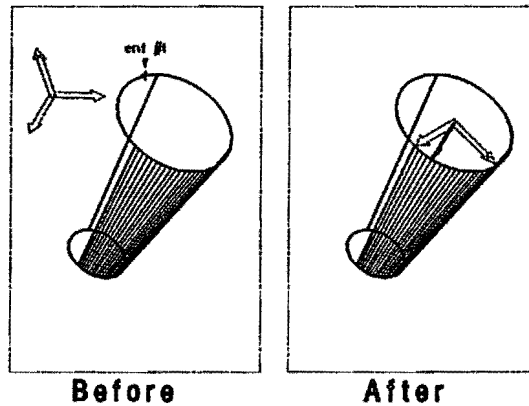


fig. (185) GCP Definition using an Arc

Removing an Image Definition

**Primary****Command:** CREATE REMIMAGE**Modifiers:** None**Command:** CREATE REMIMAGE image**Mode:** 2-D & 3-D**image:** After you select the REMIMAGE icon, the Select Image dialog shows all available image definitions. Use the pointer to select the image to remove.**Description:** REMIMAGE removes a defined image from memory.

The selected image definition is removed from memory and the image name is no longer displayed in the Image menu. Use this command to remove obsolete image definitions, allowing the creation of new, more pertinent images. The maximum number of stored images allowed at a time is 16.

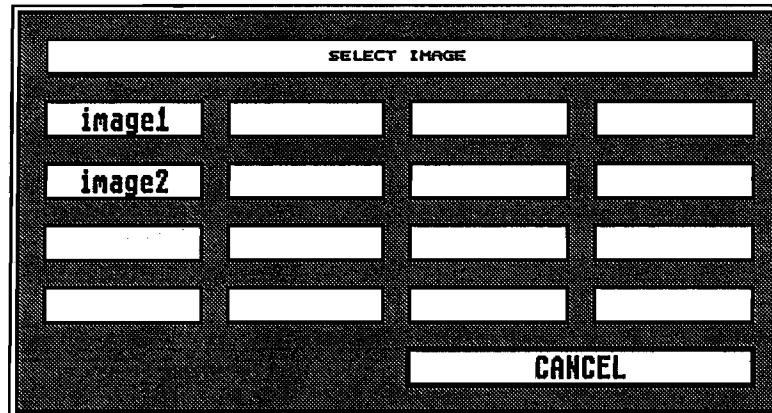


fig. (186) Selecting an Image to Delete

Creating an Image Definition



Primary

Command: CREATE DEFIMAGE

Modifiers: None

Command: CREATE DEFIMAGE name

Mode: 2-D and 3-D

name: After you select the DEFIMAGE icon, a small dialog prompts for an eight character name of the image you are creating. If you click on the Cancel button, or fail to enter a name with one or more characters, DynaCADD cancels the command.

Description: DEFIMAGE creates and stores the current page zoom and scroll factors in an image. You may recall this image at any time, from the list in the Image menu.

The current zoom factor and scrolling factors of the current drawing are stored in an image definition under the label "name". The new image name is inserted into the Image menu for later selection. When you select this image name, the screen is instantly restored to the corresponding zoom and scroll factors.

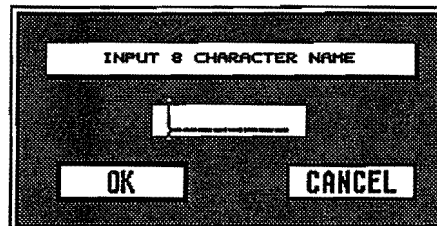


fig. (187) Input Image Name

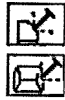
Creating a Figure

**Primary****Command:** CREATE DEFFIG**Modifiers:** None**Command:** CREATE DEFFIG ent #1...ent #n, loc #1, file**Mode:** 2-D and 3-D**ent #1...ent #n:** One or more entities, grouped and copied to a disk file. All entity types are permitted.**loc #1:** The insertion point of the selected entities. When a figure is defined and saved in a disk file, you can insert it any where on the drawing by selecting an insertion point. The two insertion points are matched when inserting the figure (the original insertion point and the point chosen when using INSERT SUBFIG or INSERT POLYFIG).**file:** Once you select the insertion point, a File Selector is displayed. Enter the filename and path for the figure. If the figure already exists, you are asked if you wish to overwrite the original file. All figure files automatically receive a .FIG extension.**Description:** DEFFIG creates and stores a group of selected entities. The figure is stored in a disk file and you may inserted it into any drawing, at a defined scale and rotation.

When you insert a figure (see Chapter 7), you have the choice of inserting it as a subfigure (all entities in the figure are grouped together and treated as one entity) or a polyfigure (all entities are separate).

All selected entities are saved to disk in the file name "file", for later recall. Location #1 is used to match with the insertion point when you insert the figure into a drawing (see Chapter 7). Once created, you can insert figures into any drawing, at any scale and rotation.

Creating Macro Definitions


Primary
Command: CREATE DEFMACRO

Modifiers: None

Command: CREATE DEFMACRO

Mode: 2-D and 3-D

Description: Use DEFMACRO to create and edit as many as 26 macro definitions. Macros are simple key stroke combinations that enable you to execute larger, more complex commands. Each macro is assigned to a single letter of the alphabet and is recalled by pressing [Ctrl] with that letter. For example [Ctrl-A].

To view all existing macros, use INFORM LIST LSTMACRO. See Chapter 14.

After you select the DEFMACRO icon, the Define Macro dialog is displayed. There are 26 key combinations available for macro assignment. To create or edit a macro:

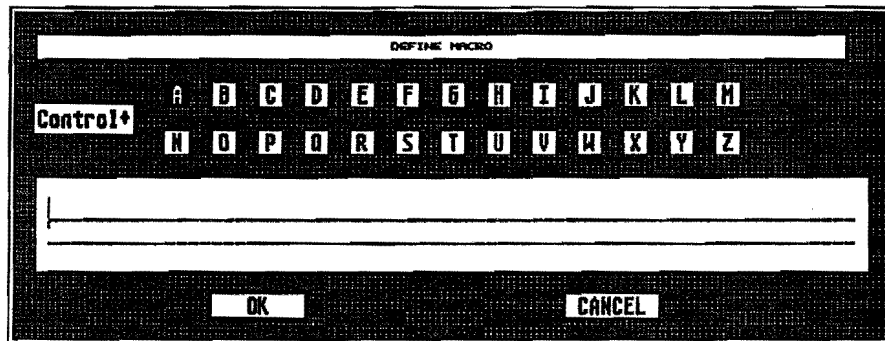


fig. (188) Defining a Macro

Tool Creation and Selection

1. Select the appropriate button from the dialog. The 26 available keys are labelled A to Z. The current macro being edited is displayed in reverse color.
2. Once you have selected the key, the current corresponding macro definition, if any, is displayed. Use the editing keys to change or create the macro to show the desired command. For example:

INSERT LINE ORTHO [space]

is a valid entry. The entry must be a DynaCADD command, written in the proper command syntax, including punctuation.

3. Click the OK button to lock any changes made to the current macro definitions.
4. Click the Cancel button to undo any changes made in the current editing session.

You can activate a macro at any time by holding down [Ctrl] and pressing the corresponding key. The text contained in the macro definition is repeated on the command line and acted upon, exactly as if the definition was typed. To make the macro definitions permanent, use the Save Macro option in the File menu.

Note:

Many commands have default attributes, visible as highlighted icons, such as LINELOC for INSERT LINE. When writing the macro, you don't need to enter the default if you want to execute that setting.

Macros cannot be chained or call each other. You cannot select entities in a macro except through BYALL [space] and BYWIN [space].

Defining the Page Origin



Primary

Command: CREATE DEFORIGIN

Modifiers: None

Command: CREATE DEFORIGIN: loc #1

Mode: 2-D and 3-D

loc #1: Selected location of the new page origin.

Description: With DEFORIGIN you can redefine the current page (2-D) origin.

As noted previously, DynaCADD has both a 2-D, or page origin, and a 3-D origin (defined by the current Geometric Coordinate Plane). DEFORIGIN defines the location of the 2-D origin only. The page origin (X0, Y0) is moved to location #1. If the Page Origin option in the Tools menu is On, the new page origin is visible, graphically represented by a small box centered on the location.

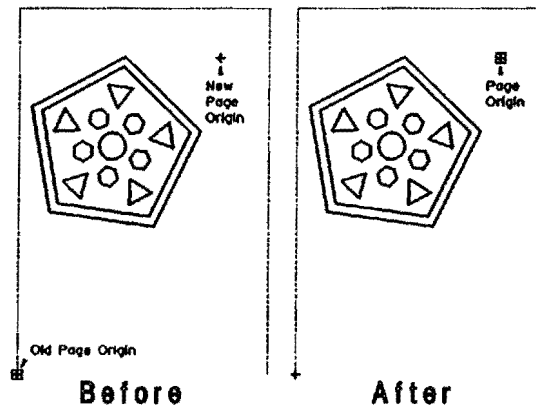


fig. (189) Defining a Page Origin

Moving the Current GCP Origin

**Primary****Command:** CREATE TRANGCP**Modifiers:** None**Command:** CREATE TRANGCP loc #1**Mode:** 3-D**loc #1:** 3-D location to which to move the origin of the current GCP.**fig:** CREATE TRANGCP**Description:** Translate (move) the current GCP.

Moves the origin of the current GCP definition. The GCP does not otherwise change attitude. See Chapter 11 for other GCP related commands.

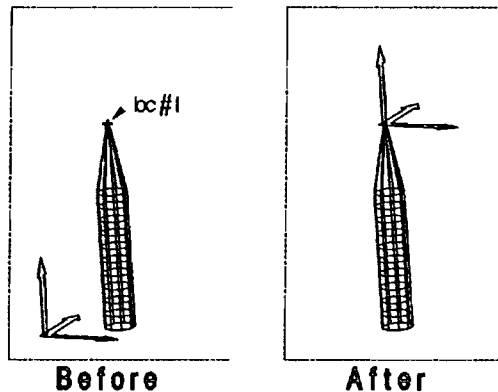


fig. (190) Moving the Current GCP Origin

Caution:

Do not translate a primary GCP. Instead, make a copy of the GCP and translate it. Follow these steps:

1. Make a view with a primary GCP or change an existing view to a primary GCP.
2. Create a new GCP in that view with **CREATE GCPVIEW** and name the GCP accordingly. For example: **Front2**.
3. Change the current GCP to the newly created GCP by selecting that name from the GCP menu.
4. Use **TRANGCP** to translate the new (current) GCP.

Output ... Chapter 13

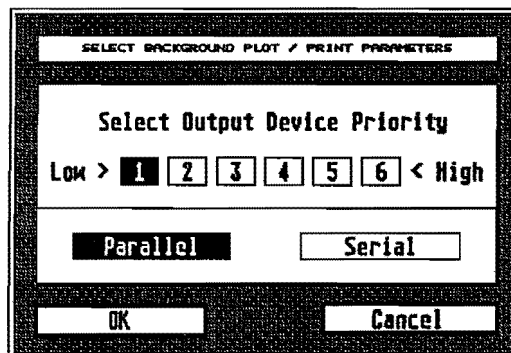
With DynaCADD you can output a full or partial drawing to a wide range of devices including plotters, printers, laser printers, PostScript devices and disk files. All output commands are combined under the OUTPUT icon. Do not confuse plotters and printers. Printers are very different devices and the drivers for one will not work for the other.

Background plot works for printing as well (see below and Chapter 5).

Background Plotting and Printing

As described in Chapter 5, DynaCADD also provides background plotting and printing; that is, you can output your drawing while continuing to work within the program, without having to wait or leave the drawing session. In order to do this, you must:

1. Click the OUTPUT icon, then click on the Plotter or Printer icon.
2. Select the plot parameters: extent, scale, paper size, pen speeds, serial or parallel port (see below).
3. Click on Disk in the Global Modifiers area. This tells DynaCADD to save the plot/print as a disk file, rather than sending it to the plotter/printer.
4. Click on Plot or Print.
5. Enter the filename into the File Selector dialog. The plot file (.PFL) is saved to disk.
6. Select Background Plot from the File menu. If you have not done so, set the device priority level (see Chapter 5) and the output port.
7. The file is output to the plotter/printer while you continue to work in DynaCADD.



*fig. (191) Background Plot/Print
Parameters Dialog*

Outputting to an External Device

**Primary****Command:** OUTPUT**Secondary****Commands:**

PLOTTER



PRINTER



POSTSCRIPT



POINTFILE

**Description:**

Output sends the current drawing to a device connected to the computer through either the serial or the parallel port, or to a disk file. The plot scale, speed, extent and size are all easily controlled as described below.

Output

Outputting to a Plotter



Command: OUTPUT PLOTTER



Mode: 2-D and 3-D

Description: OUTPUT PLOTTER sends the current drawing to a plotter connected to the computer through either the serial or parallel port. The plot scale, speed, extent and size are all easily controlled, as described below.

The RS-232 parameters should be set as follows:

Baud: 9600

Parity: None

XON/XOFF: On

RTS/CTS: Off

Bits: 8

Note: Calcomp plotter users should set RTS/CTS to On.

Setting Plot Parameters

After you select the Plotter icon, the Plotter Parameters dialog appears. If you alter the current plot parameters, you can send various portions of the drawing to the plotter, at any definable scale.

CHANGE CURRENT PLOTTER PARAMETERS											
Plotter : Roland D6 DXY-990						PEN / SPEED					
PLOT EXTENT				PLOT SCALE				1 16 6 16 B 16			
Drawing		View		None		Drawing		2 16 7 16 C 16			
Window		Dbase		User		Best Fit		3 16 8 16 D 16			
PAPER SIZE				GLOBAL MODIFIERS				4 16 9 16 E 16			
A B C D E MAX				Save		Config		5 16 A 16 F 16			
AA AB AC AD AE				Mirror		Rotate		Serial		Parallel	
A4 A3 A2 A1 A0				Inquire		Disk					
						Abort		PLOT			

fig. (192) Plotter Parameters Dialog

Selecting the Paper Size

There are 15 buttons for predefined paper size selection which refer to the plotter paper size, not the drawing size. When you select a paper size, it is advisable not to select a size larger than the mechanical limits of your particular plotter. The paper sizes are:

English sizes:

- A. 11.0 in. by 8.5 in.
- B. 17.0 in. by 11.0 in.
- C. 22.0 in. by 17.0 in.
- D. 34.0 in. by 22.0 in.
- E. 44.0 in. by 34.0 in.

Architectural sizes:

- AA. 12.0 in. by 9.0 in.
- AB. 18.0 in. by 12.0 in.
- AC. 24.0 in. by 18.0 in.
- AD. 36.0 in. by 24.0 in.
- AE. 48.0 in. by 36.0 in.

Metric sizes:

- A4. 21.0 cm. by 29.7 cm.
- A3. 29.7 cm. by 42.0 cm.
- A2. 42.0 cm. by 59.4 cm.
- A1. 59.4 cm. by 84.1 cm.
- AO. 84.1 cm. by 118.9 cm.

Other:

Max uses the plotter's maximum available size. This information is part of the plotter driver data. If Inquire is turned on and the plotter is hooked up to the serial port, DynaCADD requests the current maximum extents for the drawing size. Caution! See Inquire, below.

Selecting the Plot Scale

You can set the plot scale independent of the current drawing scale. This scale controls the size of the plot in relation to the actual size of the drawing. There are four buttons to aid in scale selection. If the drawing limits exceed the size of the selected paper or of the mechanical limits of the plotter, the drawing is clipped.

- | | |
|------------------|---|
| None: | All scaling is removed and the drawing is plotted at actual size. |
| Drawing: | The current scale of the drawing is used as the plot scale. |
| User: | After you click this button, a small dialog appears, with space to enter a definable plot scale. The plot scale is represented as a ratio of the actual drawing size. If a scale of 0.5 is selected, the drawing is plotted at one half the actual size, and a scale of 2.0 plots the drawing at twice its actual size. |
| Best Fit: | This adjusts the drawing to fit the paper size selected and plots the drawing as large as possible, insuring it does not exceed the selected paper size. |

Selecting the Plot Extents

In addition to controlling the scale at which the drawing is sent to the plotter, you can send all, or only a portion of, the current drawing to the plotter. There are four buttons for manipulating the section of drawing you wish to output:

- Drawing:** This plots the full drawing defined by the current width and height of the drawing.
- View:** This plots the extent of the currently visible drawing area.
- Window:** When you click this button, DynaCADD prompts for a window to define the portion of the drawing to plot. The request for the window does not occur until after you exit the Plotter Parameters dialog.
- Dbase:** This plots the extent of the current database. This button is usually used in conjunction with the Best Fit scale modifier.

Selecting the Plotter Port

You can attach the plotter to either the parallel or serial interface port of your computer. DynaCADD sends the plotter output to the selected port:

- Parallel:** Click this button if your plotter is currently attached to the parallel port of your computer.
- Serial:** Click this button if your plotter is currently attached to the RS-232 (serial) port of your computer. For HPGL output devices DynaCADD sends and receives data using XON/XOFF handshaking. For CalComp, and other plotters that require it, use RTS/CTS handshaking.

Global Modifiers

These options affect the entire drawing or the plot parameters.

- Save:** This option saves the current parameters to disk as the default plotter parameters.
- Config:** Select the appropriate device driver for your plotter from the File Selector dialog. If your plotter or a compatible plotter is not listed, call our Technical Support for help. You can also customize your own plotter driver by using the MAKEPLOT program, supplied with DynaCADD. The current plotter is listed at the top of the dialog.
- Mirror:** Flips the drawing on the X axis before plotting. The drawing comes out backward, useful when plotting on the bottom of mylar overlays to be used for overhead projection. Plotting on the underside of the overlay reduces light refraction when projecting the display.
- Rotate:** Rotates the drawing 90 degrees, counter clockwise, before plotting.
- Inquire:** Requests the maximum extents from the plotter. Not all plotters support Inquire. If you use this command, make sure that your plotter is turned on before you select this button; otherwise, the program waits for a response from the plotter. Use the right mouse button to abort.
- Disk:** You can send all plotter output to a disk .PFL file for later reference and background plotting. When you exit the Plotter Parameters dialog, DynaCADD displays the File Selector dialog where you enter the desired plot filename. All output is then redirected to this file.
- You can output the .PFL to the plotter while working in DynaCADD by using Background Plotting. See above and Chapter 5. The current plotter device driver must be the same as the plotter chosen for background plotting.

Selecting Pens and Pen Speed

You can control which of the pens used in the drawing are sent to the plotter and assign each pen its own individual speed, as the various pen widths available may not all plot properly at the same speed. There are sixteen pens, labelled 1 to 9 and A to F.

Selecting a Pen for Plotting

When you insert entities into a drawing, DynaCADD assigns the current pen to that particular entity. This pen number matches the plotter pen number and is synonymous with the pen color. DynaCADD only plots the entities whose pen number is selected in the Plotter Parameters dialog.

Click on the pen number to turn the pen on or off.

Selecting Pen Speed

The current speed in centimeters per second is displayed beside each of the corresponding pen numbers. To change the speed of a particular pen, click on the value and press [Esc] to clear the existing value. Enter the new speed. If you enter a value out of the range of your plotter's capabilities, the plotter selects the highest value that it can handle.

Saving the Current Configuration

Once you have selected all parameters, you can store them on disk by clicking the Save button. The saved parameters become the default parameters for each new session.

Activating the Plot

There are two buttons available for exiting the Plotter Parameters dialog:

- Plot:** Click here to send the current drawing to the plotter. The plotter's progress is displayed in a dialog. You can cancel the plot at any time by holding down the right mouse button.
- If you have Disk selected, then click on this button to see the File Selector dialog.
- Abort:** Click this button to exit the dialog without activating the plotter. Any changes made to the plot parameters are retained.

Output

Outputting to a Printer



Command: OUTPUT PRINTER



Modes: 2-D and 3-D

Description: OUTPUT PRINTER sends the current drawing to a printer connected to the computer through either the serial or parallel port. The print extent, orientation and size are all easily controlled, as described below.

Setting Print Parameters

After you select the PRINTER icon, the Printer Parameters dialog appears.

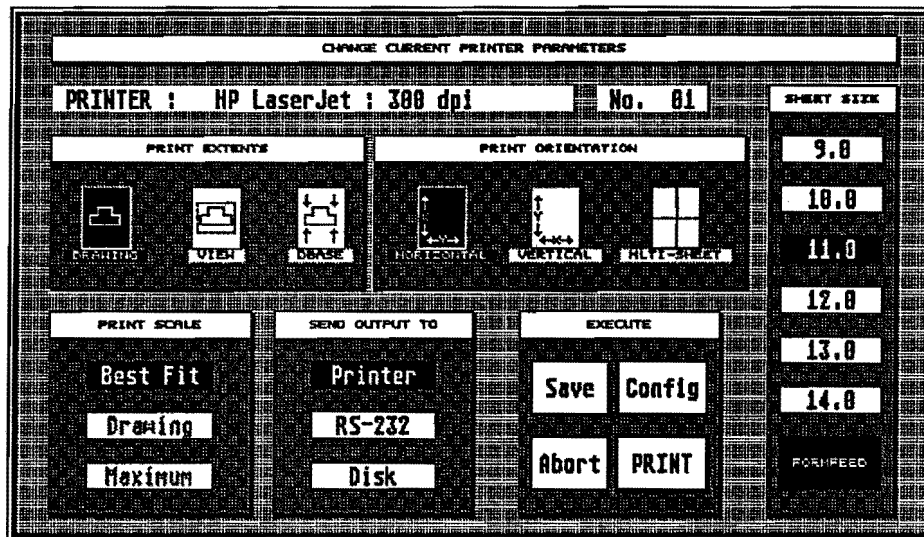


fig. (193) Printer Parameters Dialog

Selecting the Print Extents

With DynaCADD, you can send all of, or only a portion of, the current drawing to the printer. There are three buttons to manipulate the section of the drawing:

- Drawing:** This prints the full drawing defined by the current width and height of the drawing.
- View:** This prints current view in the drawing area.
- Dbase:** This prints the extent of the current database.

Selecting the Print Orientation

There are three buttons to control how the drawing appears on the printed page:

- Horizontal:** The X axis of the drawing is printed along the height of the printer paper. This is also known as "landscape mode".
- Vertical:** The X axis of the drawing is printed along the width of the printer paper. This is also known as "portrait mode".
- Multi Sheet:** The defined extent of the drawing to send to the printer is used to determine the number of sheets required. The drawing is then printed in horizontal strips, until the defined extent has been completely printed. Use this button to get full sized drawings normally only available with a plotter.
- This option is not available for use with the laser printer drivers or with the GEM .IMG drivers.

Selecting the Printer Scale

DynaCADD has three buttons to modify how it reacts to the printer's paper size. Use these buttons to scale a drawing onto the printer paper by use of a best fit:

- Best Fit:** DynaCADD scales the drawing to fit onto the printer paper.
- Drawing:** DynaCADD prints the drawing at actual size regardless of paper length. This button is always used for multi sheet prints.
- Maximum:** DynaCADD fits the width of the drawing to the width of the printer page. The height of the drawing is allowed to flow onto multiple sheets, if necessary.

Selecting the Printer Port

You can attach the printer to either the parallel or serial interface of your computer:

- Parallel:** Click this button if your printer is currently attached to the parallel port of your computer.
- Serial:** Click this button if your printer is currently attached to the RS-232 port of your computer. DynaCADD sends and receives data using either XON/XOFF or RTS/CTS handshaking, depending on the setting in the Preferences dialog.
- Disk:** You can send all printer output to a disk .PFL file for later reference. When you exit the Printer Parameters dialog, DynaCADD displays the File Selector where you enter the desired print filename. All output is then redirected to this file.

GEM .IMG files always assume output to disk.

You can output this file while working in DynaCADD by using Background Plot/Print. See above and Chapter 5. The current printer device must be the same as the printer chosen for background printing.

Selecting the Printer Sheet Size

This parameter is only considered when the Best Fit button is selected. DynaCADD insures that printing does not occur past this sheet length. There are 6 lengths available, ranging from 9 inches to 14 inches, in 1 inch increments. For paper that is 11 inches long, we recommend the 10 inch setting to prevent printing past the bottom of the page.

Final Form Feed

When printing is completed, DynaCADD sends a form feed command to your printer if the "Formfeed" button is selected. This ejects the last page.

Number of Copies

You can print multiple copies of the same drawing by entering an appropriate value into the number field. This option is available for laser printers only.

Saving the Current Configuration

Once you have selected all parameters, you can store them on disk by clicking on the Save button. The saved parameters become the default parameters for each new session.

Configuring the Printer

Select the Config button to display the list of all available printer drivers. The current printer is listed at the top of the dialog.

Output

Activating the Print

There are two buttons available for exiting the Printer Parameters dialog:

- Print:** Click this button to send the current drawing to the printer. The printer's progress is displayed in a dialog. You can cancel the print any time by holding down the right mouse button.
- Abort:** Click this button to exit the dialog without activating the printer. Any changes made to the print parameters are retained.

Outputting to a PostScript Device



Command: OUTPUT POSTSCRIPT

Mode: 2-D and 3-D

Description: Output drawing to a PostScript compatible device such as a laser printer or an imagesetter. The parameters are described on the following pages.

Setting PostScript Parameters

These parameters are the same as those described in plotting and printing, with the exception of mirror and page size.

Postscript			Encapsulated			Color		
PRINT EXTENTS			PRINT SCALE			SEND OUTPUT/GLOBAL MODIFIERS		
Drawing			Drawing			Serial Mirror X		
View			None			Parallel Mirror Y		
Dbase			Best Fit			Disk Rotate		
SET PEN / PAPER SIZE								
1	4	7	A	D	Page	EXECUTE		
2	5	8	B	E	Weight	Save		
3	6	9	C	F	No. 01	Cancel		
						PRINT		

fig. (194) PostScript Dialog

Output

Selecting the Print Extent

You can send all, or only a portion of, the current drawing to the printer by clicking on one of the three buttons below:

- Drawing:** This prints the full drawing defined by the current width and height of the drawing.
- View:** The extent of the current view is printed.
- Dbase:** The extent of the current database is printed. This button is usually used in conjunction with the Best Fit scale modifier.

Selecting the Print Scale

This controls the size of the output in relation to the actual size of the drawing. If the drawing limits exceed the size of the selected paper, or of the mechanical limits of the postscript device, the drawing is clipped.

- Drawing:** The current scale of the drawing is used as the print scale.
- None:** All scaling is removed and the drawing is printed at actual size.
- Best Fit:** This adjusts the drawing to fit the paper size and prints the drawing as large as possible, insuring it does not exceed the selected paper size.

Selecting the Printer Port

You can attach the PostScript device to either the parallel or serial interface port of your computer:

Parallel: Click this button if your PostScript device is currently attached to the parallel port of your computer.

Serial: Click this button if your PostScript device is currently attached to the RS-232 (serial) port of your computer.

Global Modifiers

These options affect the entire drawing or the print parameters.

Mirror X: Flips the drawing on the X axis before printing. The drawing comes out backward.

Mirror Y: Flips the drawing on the Y axis before printing. The drawing comes out upside down.

Rotate: Rotates the drawing 90 degrees, counter clockwise, before printing.

Output

Selecting Pens

You can control which pens used in the drawing are printed by clicking on the 15 pen buttons, labelled 1 to 9 and A to F. When you insert entities into a drawing, DynaCADD assigns the current pen to that particular entity. DynaCADD only plots the entities whose pen number is selected in the parameter dialog box.

Setting Page Parameters

- Size:** Click here to set the output page size. The range is from 0.5 inch wide and high, to 99.5 inches wide and high, in 1/2 inch increments. Click the right and left arrows to increase and decrease the size, respectively.
- No.** Click here and enter the number of copies.
- Color:** When this button is activated, all pen colors will be used. If this button is deactivated, all entities will be drawn in black.

Setup Parameters

- PS:** Click here to output in PostScript.
- EPS:** Click here to output in Encapsulated PostScript.
- Weights:** Click here to bring up a dialog to set each pen's weight. 72 units equals one point size. The default values for weights 1, 2 and 3 are 72, 144 and 216 respectively. To set a new value, select the appropriate pen number and enter the new value.

Outputting a PostScript File

There are three buttons available for exiting the Output PostScript dialog:

- Save:** Once you have selected all parameters, you can store them on disk by clicking the “Save” button. The saved parameters become the default parameters for each new session.
- Print:** Click here to send the current drawing to the PostScript device. The progress is displayed on screen in a dialog. You can cancel the plot at any time by holding down the right mouse button.
- If you have selected Disk, then click on this button to see the File Selector dialog.
- Abort:** Click this button to exit the dialog without activating the PostScript output. Any changes made to the plot parameters are retained.

Output

Outputting a Point File



Command: OUTPUT POINTFILE (file) ent #1...ent #n



Mode: 2-D and 3-D

file: Name of point file.

ent #1...ent #n: Point entities to be output.

Description: POINTFILE outputs the coordinates, X and Y (and Z in 3-D), of the selected points into a specified file. Each component of the coordinate (X,Y,Z) is separated by a comma [,]. Each coordinate is terminated by a carriage return, line feed combination (CR/LF).

Information Commands ... Chapter 14

In DynaCADD, you can query the status of virtually every aspect of your part and drawing, including:

- Detailed information on entities such as:
 - line style, weight, color, mode and type of entity, layer and entity count, X, Y, and Z point of origin, X and Y radius, rotation about the X Y plane at the point of origin, start and end angle, length along the circumference of a circle or arc, X, Y and Z rotation about the axes of the plane, types of entities in a group.
- Text height, width, slant, rotation and font
- Layers which contain entities
- Distance between two locations
- Measurements of angles, perimeters and areas
- Grid and Axis settings
- Drawing, part, GCP, layer and image names
- Drawing units, width, height and scale
- Current font
- Hatch X and Y scale
- Section angle and distance
- Image window size and corner locations
- GCP rotation and translation
- View number, scale and 2-D origin
- Current view zoom factor
- 2-D and 3-D database extents
- Current layer number, name, lock and visibility status

Information Commands

These commands are particularly useful when importing files from other formats. Also, if you are resuming a previous session, you can use Inform to examine entities such as text to make sure subsequent entities of that type are inserted with the same pen parameters.

Inform command results are displayed in the command history screen or on the printer and do not generate entities.



To display the command history and text session at any time click on the SHOWCOM icon or press [Tab].

To return to the design session you can click the windows close button or press [Tab].

The commands to measure distances or entities are similar to the Dimension commands, except that they only output the result to the text screen, not onto the drawing area.



All information commands are combined under the INFORM icon.

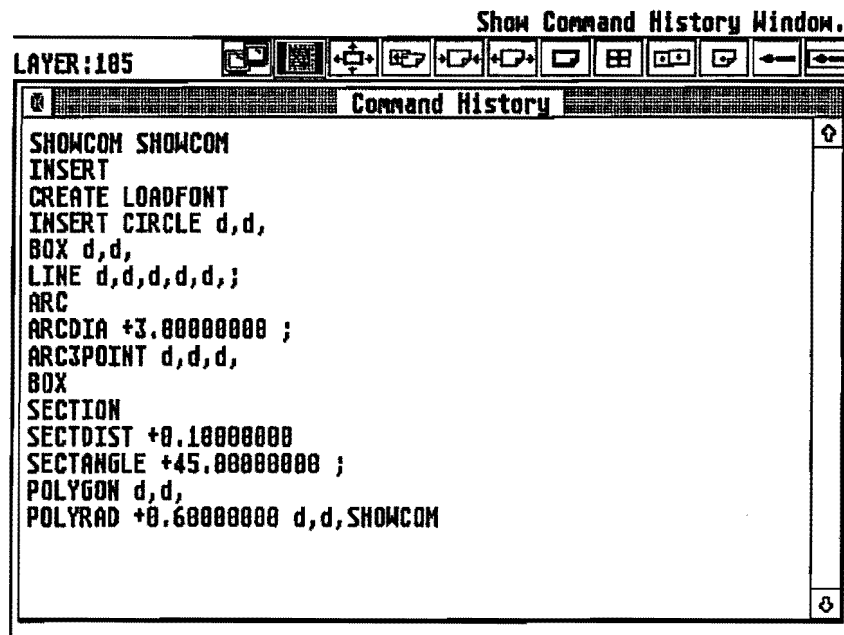


fig. (195) Command History Window

Listing Information



Primary Command: INFORM LIST

Modifiers: LSTDRAW 
LSTIMAGE 
LSTGCP 
LSTVIEW 
LSTLAYER 
LSTEXTENT 
LSTMACRO 

Description: List displays information about the selected modifier, as described below.

Listing the Current Drawing



Command: INFORM LIST LSTDRAW

Mode: 2-D and 3-D

Description: Listing the drawing displays all information related to the current drawing, including:

- drawing name
- part name
- drawing width, height and scale
- absolute Cartesian position of the 2-D origin relative to the lower left corner of the page
- drawing units

```
Command History
INFORM LIST LSTDRAW
Drawing : E:\DYNACADD\PARTS\NEW\NEW.DWG
Attached to Part : E:\DYNACADD\PARTS\NEW.PRT
WDT : 11.000000   HGT : 8.500000   SCL : 1.000000

X-Orig. : 0.000000   Y-Orig. : 0.000000
Units : Inches
Small Entity Trap : 0.077500

Large Entity Trap : 0.155015

Grid-X : 0.250000   Grid-Y : 0.250000
Axis-X : 0.250000   Axis-Y : 0.250000

Currently Active Font : Leroy Rough

Txt.Wdt : 0.050000   Txt.Hgt : 0.062500
```

fig. (196) Listing Current Drawing Parameters (a)

- small and large trap (maximum distance the selection point may be from an entity when that entity is being selected)
- grid X and Y increments
- axis X and Y increments
- current font
- text width, height, slant and rotation
- hatch X and Y scale
- section angle and distance

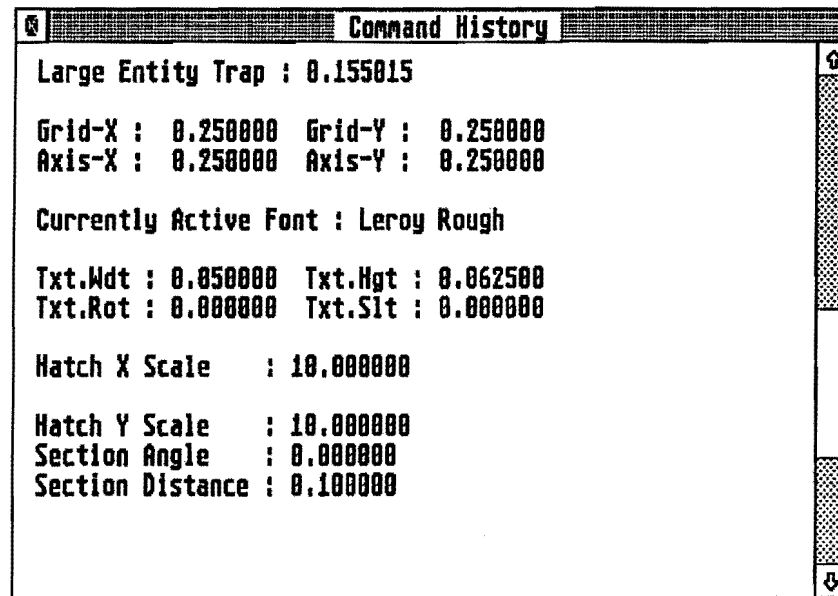


fig. (197) Listing Current Drawing Parameters (b)

Listing All Defined Images



Command: INFORM LIST LSTIMAGE

Mode: 2-D and 3-D

Description: LIST LSTIMAGE displays information relating to all previously defined images, including:

- image name, given when created
- X1, Y1 and X2, Y2: two diagonally opposing locations that define the image window.

```
Command History

INFORM LIST LSTIMAGE

IMAGE Name : Overview
X1 :-8.000000      Y1 :-8.000000
X2 : 12.943769     Y2 : 8.500000

IMAGE Name : Frame
X1 : 2.750000      Y1 : 2.125000
X2 : 9.221884      Y2 : 6.375000

IMAGE Name : 3d ent
X1 : 3.228042      Y1 : 2.680471
X2 : 8.224572      Y2 : 5.961626
```

fig. (198) Listing All Defined Images

Listing All Geometric Coordinate Planes



Command: INFORM LIST LSTGCP

Mode: 2-D and 3-D

Description: LIST LSTGCP displays information relating to all previously defined GCPs, including:

- GCP name, given when created
- the rotation about the X, Y and Z axes, in degrees
- the X, Y and Z axis translation (the distance of the GCP from the absolute origin - X0, Y0, Z0)

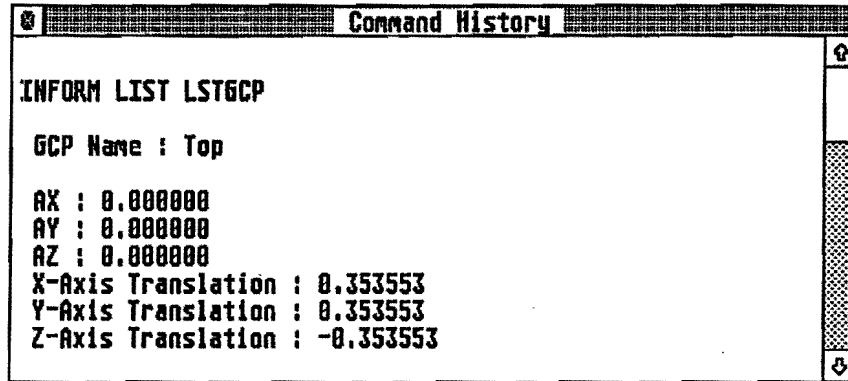


fig. (199) Listing All Geometric Coordinate Planes

Listing All Active Views



Command: INFORM LIST LSTVIEW

Mode: 2-D and 3-D

Description: LIST LSTVIEW displays all information related to the all active views, including:

- view number
- 2-D origin of the view
- the lower left and upper right hand 2-D locations of the view clip
- X and Y distances the view has been scrolled from the view origin
- view scale
- current view zoom factor
- rotation in degrees about the X, Y and Z axes
- primary GCP

```
Command History
INFORM LIST LSTVIEW

View : 1
Pg.X : 5.000000    Pg.Y : 4.000000
X1 : 2.750000     Y1 : 2.125000
X2 : 8.250000     Y2 : 6.375000

X-Scr :-1.463322   Y-Scr :-1.169535
Scale : 0.785000   Zoom : 0.938348

Axis X Rotation : 12.952540
Axis Y Rotation : 329.132522
Axis Z Rotation : 7.630740

Primary GCP : Front
```

fig. (200) Listing All Active Views

Listing Working Layers



Command: INFORM LIST LSTLAYER



Mode: 2-D and 3-D



Description: LIST LSTLAYER displays all layers that contain at least a single entity, as well as:

- layer number
- number of each entity type in the layer
- layer name (if any)
- current state (visibility) of the layer - on or off
- if the layer is locked or unlocked

See also Inform Count, below.

Command History										
INFORM LIST LSTLAYER										
LAY	POI	LIN	CIR	ARC	ELL	ELA	TXT	BSP	BEZ	SOL
---	---	---	---	---	---	---	---	---	---	---
0	0	3	0	1	0	0	0	0	0	0
2	0	21	0	0	0	0	0	0	0	0
255	0	4	0	0	0	0	0	0	0	0
0	:				:	ON	:	UNLOCKED		
2	:	ARTIFICAL_GCP			:	ON	:	LOCKED		
255	:				:	ON	:	UNLOCKED		

fig. (201) Listing Working Layers

Listing Database Extents



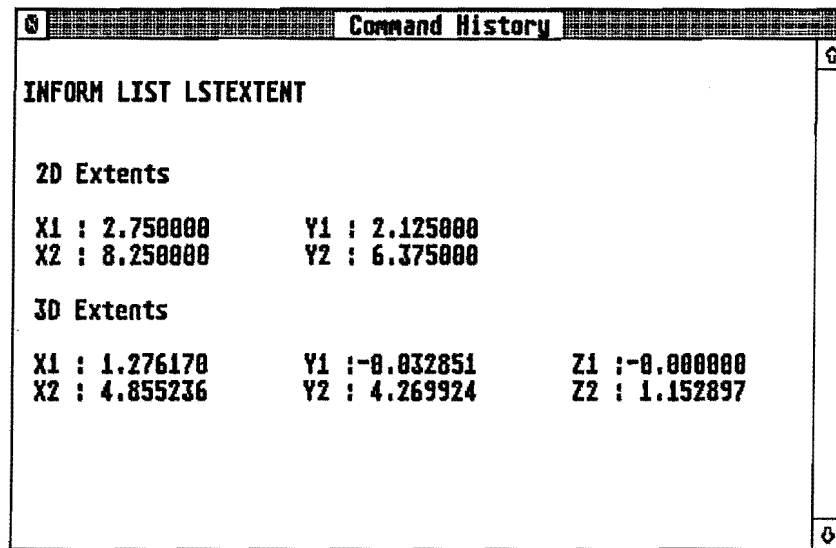
Command: INFORM LIST LSTEXTENT



Mode: 2-D and 3-D



Description: LIST EXTENT displays the 2-D and 3-D extents of the current part and drawing. The extents are defined as the maximum coordinates used by the entities in the database, displayed as the most distant locations on the X and Y or X, Y and Z axes.



```
Command History

INFORM LIST LSTEXTENT

2D Extents
X1 : 2.750000      Y1 : 2.125000
X2 : 8.250000      Y2 : 6.375000

3D Extents
X1 : 1.276170      Y1 : -0.032851      Z1 : -0.000000
X2 : 4.855236      Y2 : 4.269924      Z2 : 1.152897
```

fig. (202) Listing Database Extents

Listing Macro Definitions



Command: INFORM LIST LSTMACRO



Mode: 2-D and 3-D



Description: LIST LSTMACRO displays all defined macros, with the key combination followed by the actual macro definition. For example:

Control +A :

INSERT LINE LINELOC

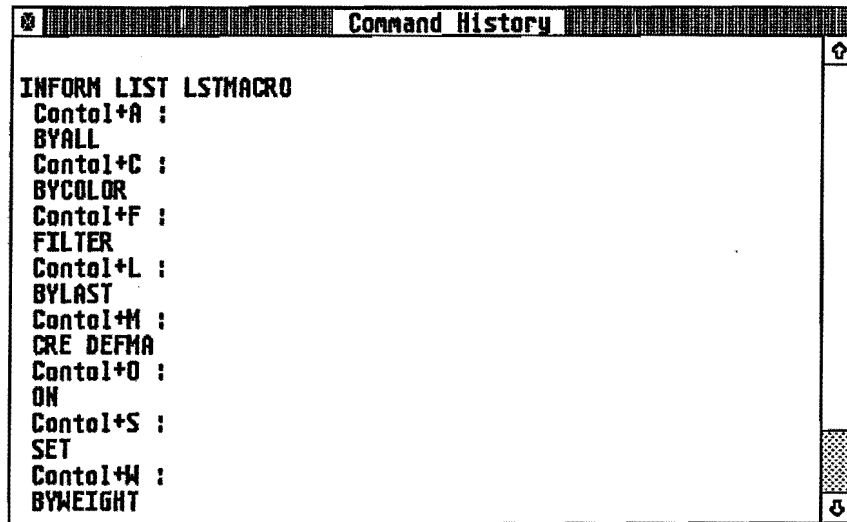


fig. (203) Listing Macro Definitions

Counting Entities

**Primary****Command:** INFORM COUNT**Modifiers:** None**Command:** INFORM COUNT**Mode:** 2-D and 3-D**Description:** COUNT displays the current number of entities.

COUNT displays the individual entity counts as well as the total entity count, for all layers. See also INFORM LIST LSTLAYER, above.

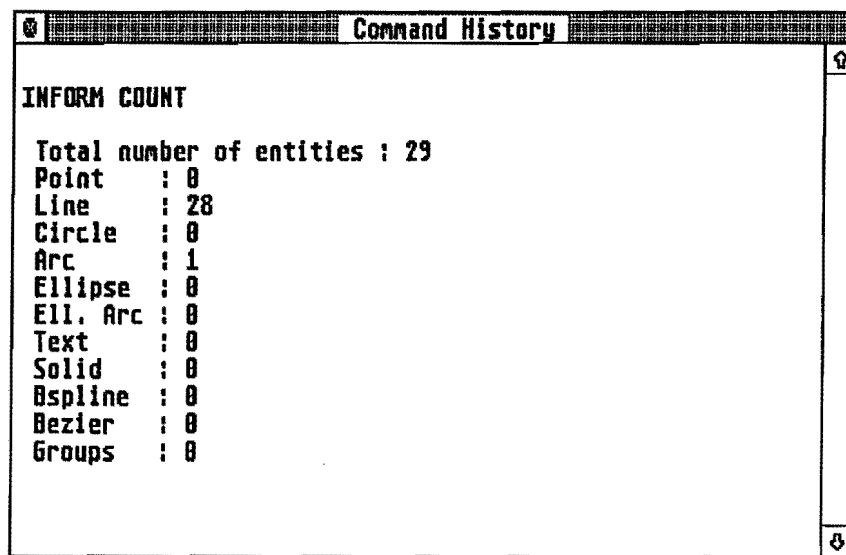


fig. (204) Counting Entities

Verifying Entities



Primary

Command: INFORM VERIFY

Modifiers: None

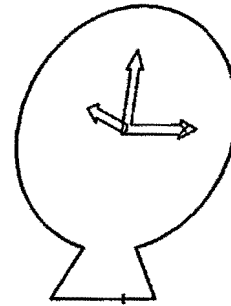
Command: INFORM VERIFY: ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: One or more selected entities to have their individual statistics listed.

Description: VERIFY displays statistics relating to the selected entities, depending on the entity type, including:

- entity type and mode
- pen number (color)
- line style name
- line weight
- layer



ent #1

```

Command History

INFORM VERIFY ;

Entity Type : 3D Line
Color : 1
Style : CONTINUOUS
Weight : 3
Layer : 0
X1 : 2.199678      Y1 :-0.032851      Z1 :-0.000000
X2 : 3.931728      Y2 :-0.032851      Z2 :-0.000000
Line Length : 1.732051
    
```

fig. (205) Verifying Entities (a)

Information Commands

- the X1, Y1 (and Z1 in 3-D) point of origin
- the X and Y radius
- rotation about the X-Y plane at the point of origin
- start angle
- end angle
- length along the circumference of the circle or arc (not ellipse or elliptical arc)
- X and Y and Z (3-D only) rotation about the axes of the plane on which the entity sits.

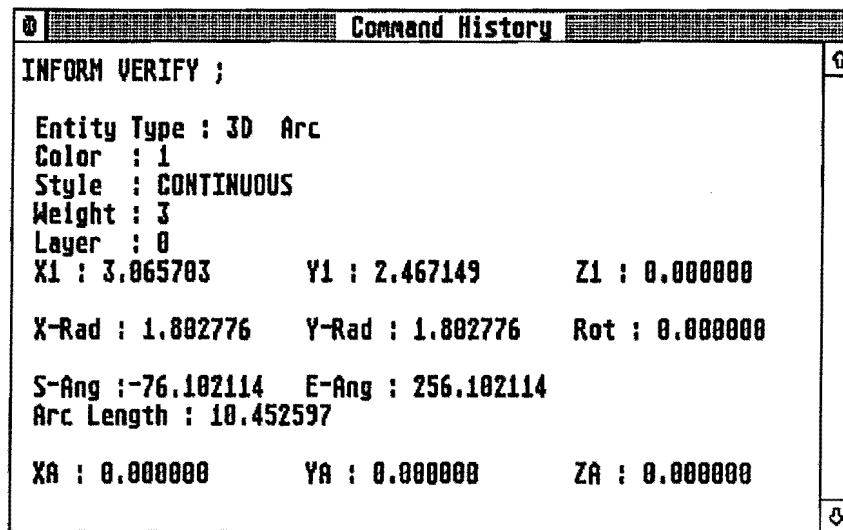
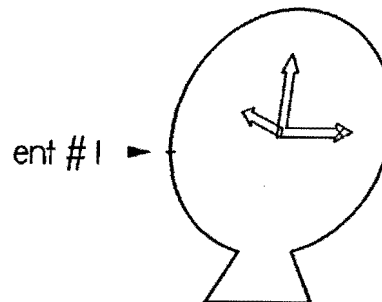


fig. (206) Verifying Entities (b)

Measuring



**Primary
Command:**

INFORM MEASURE

Modifiers:

MEASANGLE



MEASDIST



MEASPERIM



MEASAREA



Description:

Measure calculates and displays the selected parameter.

Measuring Angles



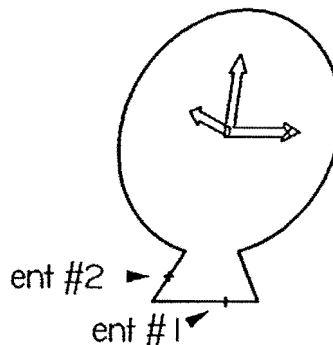
Command: INFORM MEASURE MEASANGLE: ent #1, ent #2



Mode: 2-D and 3-D



ent #1, ent #2: Two selected lines used as references for the measurement. The angles between the two entities are displayed.



Command History	
INFORM MEASURE MEASANGLE	
Line 1 :	60.00000000 Degrees 1.04719755 Radians 60-0'0.00000"
Line 2 :	0.00000000 Degrees 0.00000000 Radians 0-0'0.00000"
Angle 1	60.00000000 Degrees 1.04719755 Radians 59-59'60.00000"
Angle 2	300.00000000 Degrees 5.23598776 Radians 300-0'0.00000"

fig. (207) Measuring Angles

Description: The angle and the angle of each of the two selected lines are displayed, in decimal degrees, radians and degrees/minutes/seconds. In 3-D mode, the current GCP is used to define the X-Y coordinate plane. The two lines are projected on to this plane, and the angle is measured relative to that plane.

The angle of each line is displayed, followed by the two angles created on each side of their intersection. If the two lines do not actually meet, DynaCADD projects their ends to a point where they could meet, if trimmed, and figures the angles at that point.

Measuring Distances



Command: INFORM MEASURE MEASDIST: loc #1, loc #2

Mode: 2-D and 3-D

loc #1, loc #2: Two selected locations defining the distance to measure.

Description: The location of each of the two points and the distance between the two locations are displayed. In 3-D mode, the 3-D distance is displayed.

- loc #1: X1 and Y1 points (Z1 in 3-D)
- loc #2: X2 and Y2 points (Z2 in 3-D)
- distance between each point in each axis
- 2-D distance between locations

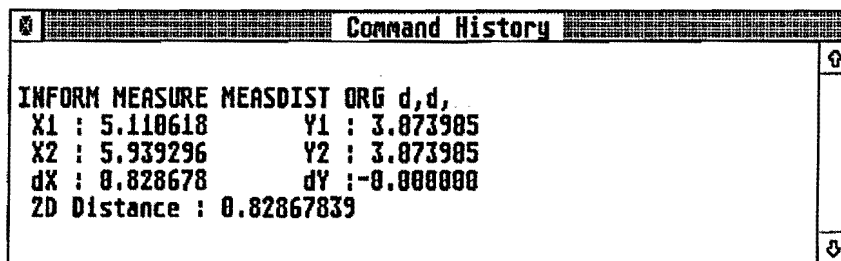
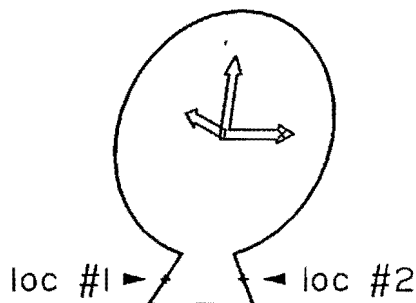


fig. (208) Measuring Distances

Measuring Perimeters



Command: INFORM MEASURE MEASPERIM: ent #1...ent #n

Mode: 2-D and 3-D

ent #1...ent #n: One or more lines, circles, arcs or fillets.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

The total perimeter of all selected entities is displayed. If all entities are contiguous, and are selected in that order, DynaCADD displays this information, otherwise it displays a "Non-contiguous perimeter" message.

The perimeter is displayed in the text screen.

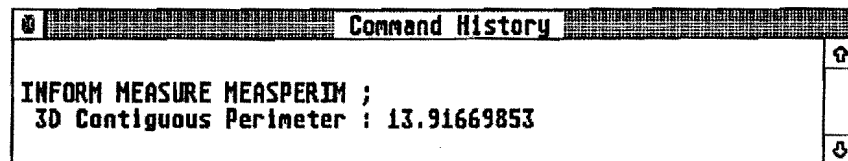
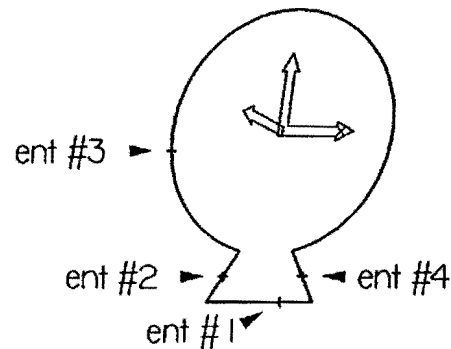


fig. (209) Measuring Perimeters

Measuring Areas



Command: INFORM MEASURE MEASAREA ent #1...ent #n



Mode: 2-D and 3-D



ent #1...ent #n: The contiguous entities which enclose the area to measure.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

Measures the area of any enclosed entities (composed of lines, arcs or circles including irregular polygons). Multiple entities must be contiguous and completely enclose the area to be measured.

The area is displayed on the text screen in units squared.

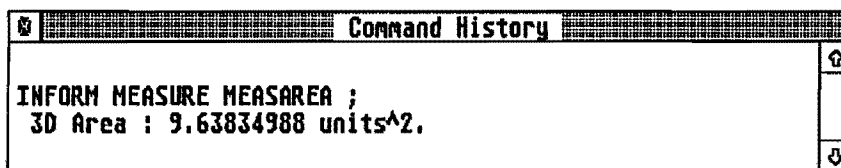
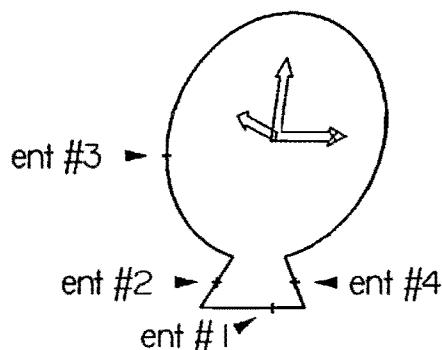


fig. (210) Measuring Areas

Layers ... Chapter 15

A drawing is made up of 256 layers. In 2-D mode, layers are like sheets of paper stacked one on top of another. In 3-D mode, the sheet also extends into the Z axis, like a wrapper rather than a flat sheet. Each of which may have its transparency turned on or off, or be locked from modification. Each layer is numbered in sequence, from 0 to 255. You can also name a layer and refer to it interchangeably by its name or by its number.

When entering a range of layers, you can enter single numbers or names, a range of layers, or a combination of both. Names and numbers can be combined interchangeably. For example, these are all valid range entries:

32-65, 85, 96-102	all layers from 32 to 65 inclusive, plus layer 85, plus all layers from 96 to 102 inclusive.
test-12	all layers from layer test to layer 12, inclusive.
1, 2, 5, 7, 255	the indicated layers only.
0-255	all layers.
test1, test2, test3, 7, 9, 11	the indicated named and numbered layers.
61-argus, test2-test9, part1-12	layers 61 to layer argus inclusive, plus layers test2 to test9, inclusive, plus layers part1 to layer 12, inclusive.

The name or number of the current layer appears at the upper left corner of the drawing area when the pointer is not over an icon. A layer name can have up to 17 characters for DXF file compatibility with only 15 characters displayed.

When a DXF file is imported into DynaCADD, all layer information is retained from the transferred file.

All layer editing and modification commands are combined under the EDITLAYER icon.

Editing Layers



Primary

Command: EDITLAYER

Modifiers:

SELLAYER



SELLAYERT



LAYERON



LAYEROFF



LAYLOCK



LAYUNLOCK



LAYERNAME



COPYLAYER



COLLAYER (flag)



COLSTYLE (flag)



COLWEIGHT (flag)



Flags:

The flags control how the entities are displayed. Every layer has a set of attributes assigned to it. When the appropriate flag is enabled, the entities will be displayed according to the attribute controlled by that flag. When it is disabled, the entities will be displayed with their original attributes. The attributes can be set using the Layers Dialog in the Set menu. For more information, see Chapter 5.

COLLAYER The pen color.

COLSTYLE The pen style.

COLWEIGHT The pen weight.

Description:

Manipulate the layers through a variety of commands as described below.

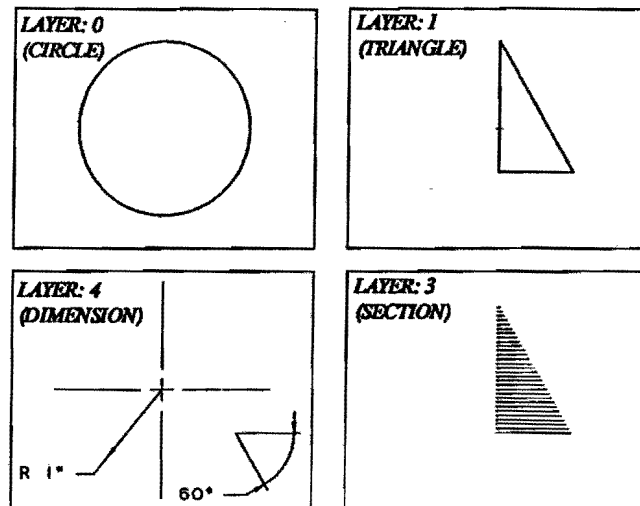


fig. (211) Layer Sample Names and Numbers

Selecting Layers



Command: EDITLAYER SELAYER lay



Mode: 2-D and 3-D

lay: After you click the SELAYER icon, enter a layer number (0 to 255) or a layer name in the dialog box, to select that layer. Only one layer can be entered here, not a range.

Description: The selected layer becomes the active (current) layer. Any new entities are created on this layer. SELAYER also turns off all other layers (see LAYEROFF, below), regardless of their on/off status. Only entities on the selected layer are visible.

If you select a locked layer as the current layer, it is automatically unlocked (see LAYLOCK and LAYUNLOCK, below).

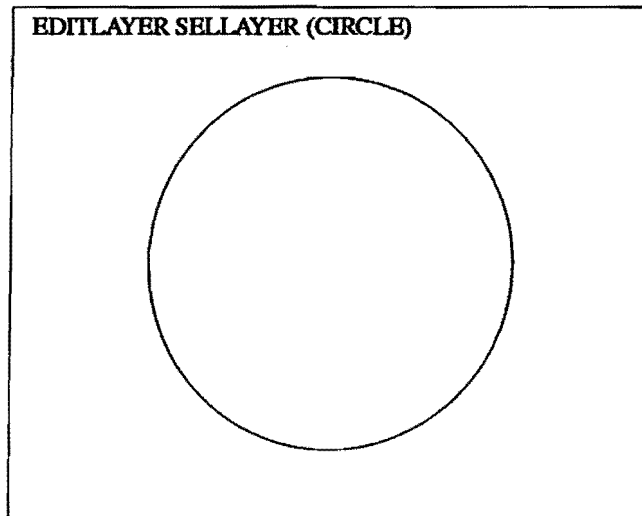


fig. (212) Select Active Layer

Selecting Layer Transparent



Command: EDITLAYER SELLAYERT lay



Mode: 2-D and 3-D

lay: After you click the SELLAYERT icon, enter a layer number (0 to 255) or a layer name in the dialog box, to select that layer. Only one layer can be entered here, not a range.

Description: The selected layer becomes the active layer. Any new entities created are automatically installed on this layer. Unlike SELLAYER (above), SELLAYERT does not affect the transparency of any other layers. The on/off status of other layers is unchanged.

If you select a locked layer as the current layer, it is automatically unlocked (see LAYLOCK and LAYUNLOCK, below).

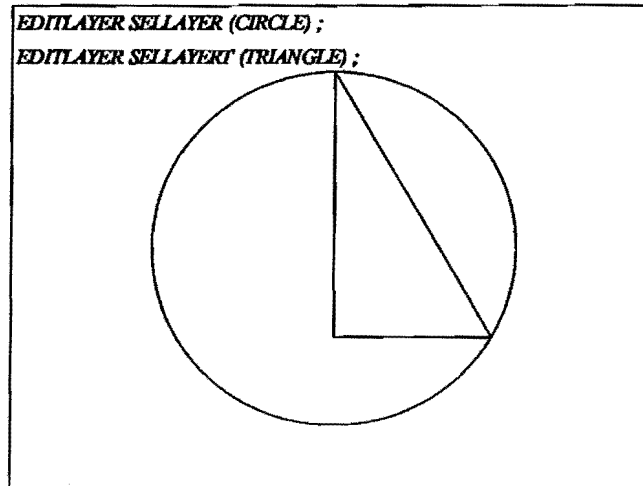


fig. (213) Select Layer Transparent

Selecting Visible Layers



Command: EDITLAYER LAYERON lay



Mode: 2-D and 3-D

lay: A selection or range of layers to make visible.

Description: LAYERON makes the selected layers visible. This is the opposite of LAYEROFF, below. The current layer is always on. Entities on all layers turned on are visible and entities on layers not turned on are invisible.

Unless they are on a locked layer (see LAYLOCK, below), you can modify (transform) all visible entities.

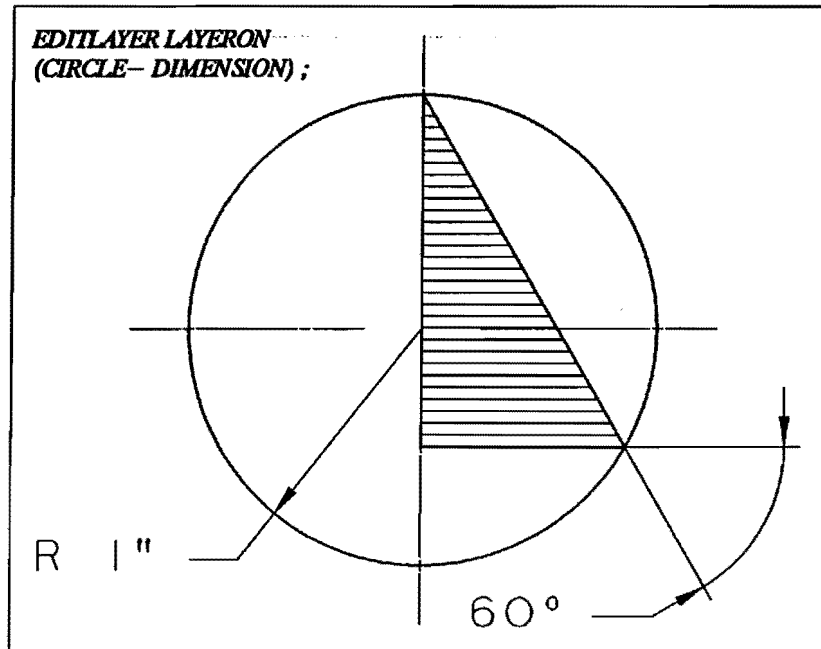


fig. (214) Turn Layers On

Selecting Invisible Layers



Command: EDITLAYER LAYEROFF lay

Mode: 2-D and 3-D

lay: A selection or range of layers to make invisible.

Description: LAYEROFF makes the selected layers invisible. This is the opposite of LAYERON, above. The current layer is always on and cannot be turned off. Entities on all layers turned on are visible and entities on layers not turned on are invisible.

You cannot modify (transform) entities on invisible layers.

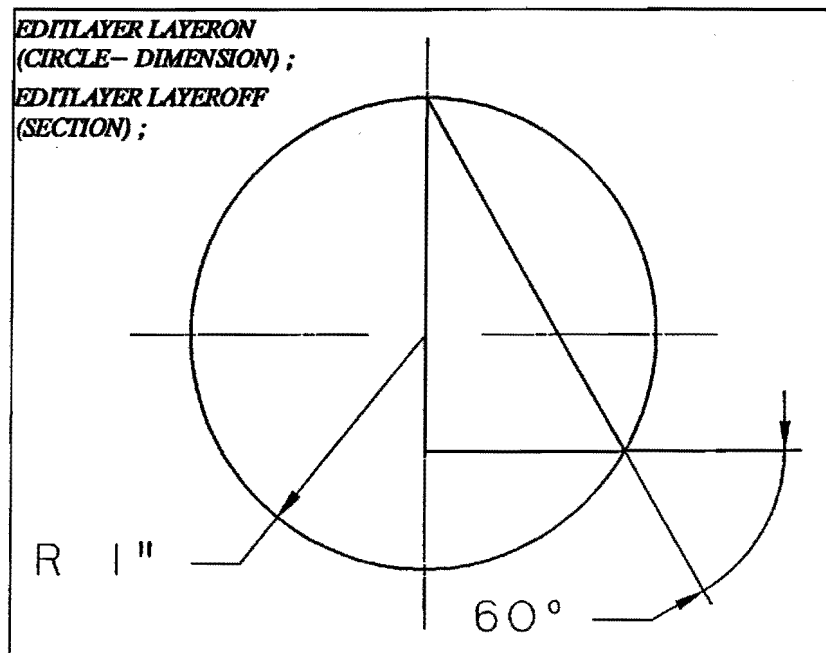


fig. (215) Turn Layers Off

Layers

Locking Layers



Command: EDITLAYER LAYLOCK lay



Mode: 2-D and 3-D

lay: A selection or range of layers to lock.

Description: You cannot edit or delete entities on locked layers, even if visible. You cannot lock the current layer.

Locked layers remain visible but cannot be modified. If you copy entities from an unlocked layer to a locked layer, they become locked (see COPYLAYER, above).

Unlocking Layers



Command: EDITLAYER LAYUNLOCK lay



Mode: 2-D and 3-D

lay: A selection or range of layers to unlock.

Description: This unlocks layers so that the entities can be modified. The current layer is always unlocked.

Naming Layers



Command: EDITLAYER LAYERNAME lay



Mode: 2-D and 3-D

lay: Name assigned to the current layer.

Description: Every layer has a number (0 to 255), and can also have a name. To enter or edit a name, press [Esc] to clear any existing entry and type a new name. Click OK. To clear an existing name, press [Esc] and click OK without entering a new name.

*fig. (216) Layer Name
Dialog*

Layer :

Copying Entities to Another Layer



Command: EDITLAYER COPYLAYER lay ent #1...ent #n



Mode: 2-D and 3-D

lay: After you click the COPYLAYER icon, enter a layer number (0 to 255) or name to which to copy the selected entities.

ent #1...ent #n: One or more entities to copy from the current layer to the selected layer. All entity types are permitted.

Description: Activate the command by clicking the right mouse button, or by pressing [;].

All selected entities are copied from the current layer onto the selected layer. The original entities are not modified. The copies assume all attributes of the originals, except for the layer number.

The TRANCOPY icon does not affect COPYLAYER; a duplicate of the entities is always made.

If you copy entities to a locked layer, the copies are also locked from modification (see LAYLOCK, below).

COPYLAYER stays active while using other layer commands. For example, you can turn one layer on, select entities, then turn that layer off and turn another on, and so on.

INPUT LAYER(S) BY NAME OR INDEX 1 XXX,XXX-XXX

Layer Range :

OK Cancel

fig. (217) Layer Range Entry Dialog

Vector Font Editor ... Chapter 16

Introduction

Most computer software uses “bit-mapped” fonts. Typically these are drawn to suit one fixed size, then expanded or reduced according to the display commands of the software. With a fixed size font, the individual pixels are drawn in a fixed raster, the size of which determines the size of the character on the screen. If you want to expand or reduce a fixed size character, you can only do so by inserting or deleting complete sets of points, which often ruins the look of the character.

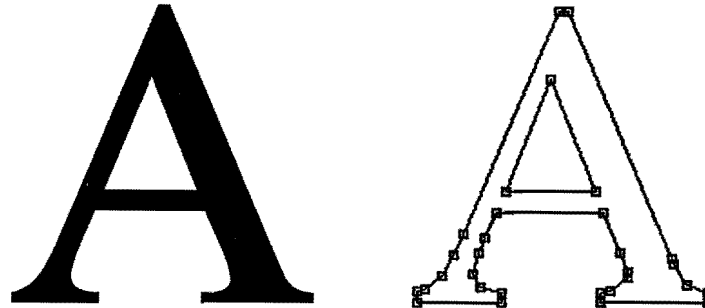


fig. (218) Bit Mapped versus Vector

Because DynaCADD uses vector fonts, it can perform fast, smooth transformations to fonts or, by entering new values, wholly recast them. The major benefit of vector fonts is obvious: you can show every character as originally defined, at any size.

In a vector font a character is defined by a series of coordinate, or control points, connected by lines. Only a few points need be defined to create a character. The actual number of points you need depends on the complexity of the character involved: DynaCADD vector fonts can have up to 64000 points on a side. DynaCADD vector fonts are not filled, or solid.

The Vector Font Editor was created to let you easily develop new, personal fonts or characters, as well as editing existing character sets.

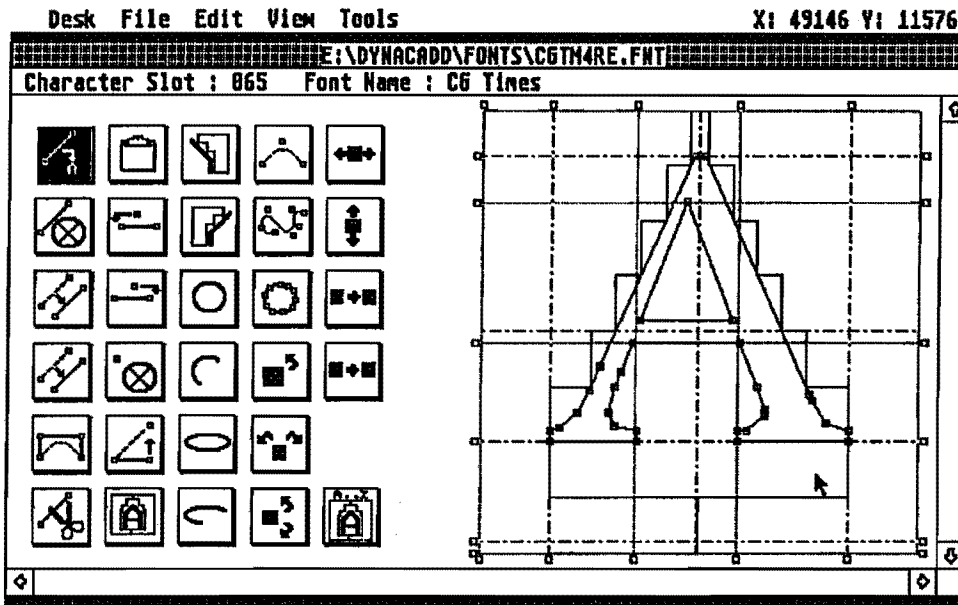


fig. (219) DynaCADD Vector Font Editor

The editor uses several drawing tools, including circle, arc, ellipse, Bézier curves and B-spline curves, which create smoothly curved vectors.

Vector fonts are most commonly used for output with plotters. In order to output a vector font character, DynaCADD calculates the location of all the points, then draws the lines to connect them. The result is that you can recalculate a character and display it to almost any size and shape. In DynaCADD, since the internal resolution is high, the output quality depends solely on what output device you use.

While it is easy to design characters and symbols, creating an entire font takes considerable time and effort. It is sometimes easier to edit an existing font than to create a new one from scratch. This manual can only teach you how to use the font editor, but typography is an old art, and many books are available to learn about fonts and font design criteria.

To run the vector font editor, type FONTEDIT at the command prompt.

Fonts, Characters and Vectors

A font, or character set, is a single group, consisting of up to 255 characters, in a particular typeface. A typical font contains all of the letters, numbers, punctuation marks, special characters, ligatures, etc., in one version of a typeface. Normally, when one talks of a particular font, a specific size is intended. Since DynaCADD fonts are scaled within the program, size is not an issue.

In DynaCADD, a font can also be any collection of special characters, such as electrical or electronic symbols, mathematical symbols, etc. However, each symbol or character corresponds directly to a character in the system character set. For example, if you replace the letter [A] with a symbol of a transistor, when you load the symbol font and press [A] on the keyboard, the transistor symbol appears on the screen instead.

When you create a new font, you have a "palette" of 255 positions or slots in which you can define characters. You can define as many or as few as you wish, or even add new characters to or remove characters from a font, as long as the 255 character limit is not exceeded.

A font can combine symbols and special characters with regular letters, numbers and so on. Characters outside the regular numbers, letters and punctuation marks can be displayed in DynaCADD by pressing combinations of keys, such as [Alt] and [Ctrl].



!"#\$%&'()*+,-./012345678
 9:;=<?@ABCDEFGHIJKLMNO
 PQRSTUVWXYZ[\]^_`abcdefghijklmnopqrstuvwxyz{|}~Çüé
 âãäåçêëèìíîïÏÄÅÉæÆôöòûüÿ
 ÖÜ££βƒáíóúñÑªº¿†”“„¡«»ãõ
 ØœÀÃÖ”†¶©® ¡¡¡

fig. (220) Character Set with Symbols

A character consists of one or more vectors. A vector is any group of control points (up to 127 control points maximum in a single vector) connected by lines. A character can be as complex as you wish, within the limits of the drawing window (64000 points on a side).

The vector is the key element in creating a character. Imagine a vector as a piece of string and the control points as knots in it. The knots determine the complexity of the character and the smoothness of the curves. There must be a specific start and end point (knot) for each piece of string. A character can be built from any number of pieces of this "string". And, like string, vectors can be cut into smaller pieces. Vectors, however, cannot be joined together.

When you create a new character, you do so by drawing each vector: positioning the control points (the "knots"), one at a time. As you add points (by clicking the mouse button), a line appears, connecting the pointer with the last control point. This line stretches with the pointer and is anchored in place when you set another control point.

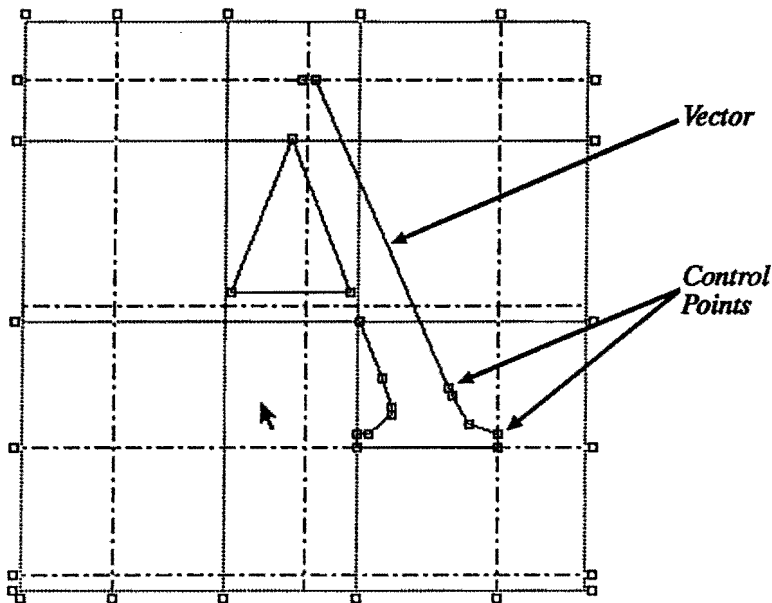


fig. (221) Drawing Window with Vectors and Control Points Labelled

When you press [Return], that vector is “set” with all the control points you entered at their current locations. If you do not press [Return] when you select another tool, the vector does not remain on the screen and is lost for future use. Control points can be added to, deleted from or moved to a new location in a vector, or the entire vector can be moved, copied or deleted.

Complexity is a consideration when creating a font, especially when the output device is a plotter which outputs a character a vector at a time. The more, separate vectors a character uses, the longer it takes to draw. If possible, use fewer vectors and make continuous connections.

Buffers

Buffers are reserved memory storage areas. Their use is very important in the font editor. You can save entire characters with the copy command, or save individual vectors with the copy vector command. Once saved, characters or vectors can be copied into the same or other characters with a few simple commands.

There is only one character buffer. Anything copied into it overwrites the current contents. Before editing a character, it is a smart idea to copy the character into the character buffer, so that you can restore the original if you make mistakes. Click on Copy or press [Ctrl-C]. To restore the contents of the buffer, press [Ctrl-V]. Be careful: if you change to a different character from that stored in the buffer, pressing [Ctrl-V] overwrites the current character with the buffer contents.

Edit	
Undo	⌘Z

Cut	⌘X
Copy	⌘C
Paste	⌘V

✓ Helplines	
✓ Guides	
✓ Cpoints	
✓ Kerning	

The Cut ([Ctrl-X]), Copy ([Ctrl-C]) and Paste ([Ctrl-V]) commands listed in the Edit menu are full character commands, not single vector commands. Vector commands are listed on the control panel. Use [Ctrl-X] to clear a character completely from the drawing window.

Note: The character buffer cannot be saved to disk.

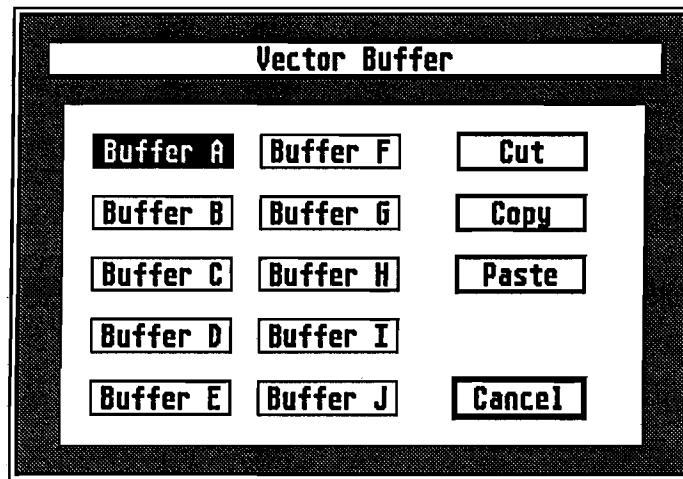


fig. (222) Vector Buffer Dialog

Vector buffers are somewhat different from the character buffer. There are ten separate buffers (A to J), which can each store a single vector. The entire group of ten buffers can be saved on disk, or loaded into memory, using the load and save buffers commands, in the File menu.

When you click on Vector Buffers, you see a dialog with the buffer names (A to J). The selected buffer is highlighted. Click on the letter button to select another buffer. The three vector buffer operations, cut, copy and paste, are listed on the right of the dialog. Select one operation such as copy. When you return to the drawing window, click on any control point of the vector to cut or copy. Cut vectors are removed from the drawing window, copied vectors are not. Both commands retain the vector in the selected buffer.

Paste brings the vector from the buffer to the drawing window. Move the vector with the mouse to the desired location and press the left mouse button. A vector in a buffer can be pasted any number of times into the drawing window by pressing the mouse button for each paste. To leave the paste mode, select another vector operation command.

You can use the vector buffers to move vectors between characters or make multiple copies of vectors. Like characters, it is a good idea to copy a vector into a buffer before you start changing it, so you can restore it later, in case you change your mind or make a mistake.

Creating and Editing a Character

If you are creating a new font, then you should follow these basic steps:

- Make sure no font is currently in memory.
- Choose Select Character from the File menu.
- Choose the character to create.
- Select the vector command and start drawing.

A new font has a “palette” of 255 positions or slots in which you can define characters. You can define as many or as few as you wish, or even add new characters to a font, as long as the 255 character limit is not exceeded.

If you are editing an existing font, then your first step is:

- Open a font file from the disk and then follow the steps mentioned above, from choosing “Select Character” to selecting the vector command.

In both cases, save your font after each character you design or edit.

Selecting a Character

Before you create or edit a character, you must select on which one you want to work. The Select Character option, in the File menu, displays the system character set. This display represents the character slot to edit and shows how the system displays that particular character. You do not select a system character to edit, only the slot that character occupies.

Since only a few characters can be displayed at a time, click on the page Left and Page Right buttons to move to the appropriate part of the display. To select a character slot to edit or to create, click on the system character you want to replace.

The character currently in the drawing window is highlighted. The buttons for character slots with characters in them are shadowed.

To display another character in the font, use the Select Character option in the File menu. You can also press [+] to move up one character, e.g. from A to B, or press [–] to move down a character, e.g., from M to L. If you press [–] when at character slot 1, you go backward to character slot 255.

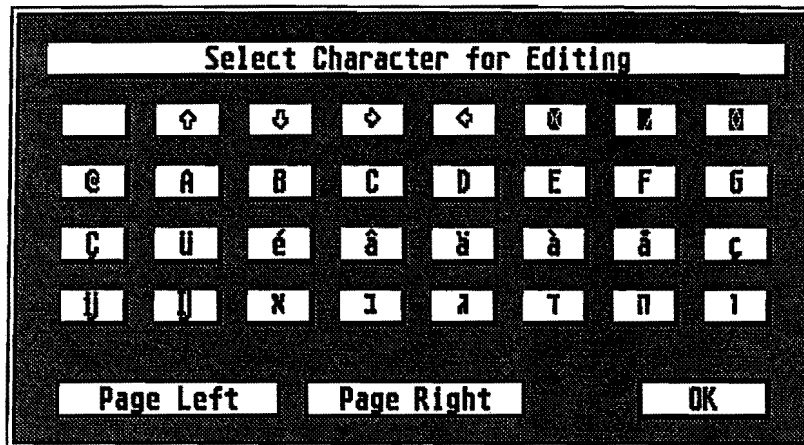


fig. (223) Character Selection Dialog

The characters are numbered according to the widely used American Standards Code for Information Interchange - ASCII system. Although the ASCII character set includes a character numbered zero (not the character "0", which is actually ASCII number 48), you cannot change it in this font editor. The character numbers that can be changed range from 1 to 255. "A", for example, is number 65, "B" is 66, "a" is 97, "b" is 98, and so on. In the ASCII numbering system, every character has its own number. See the appendix for a chart of ASCII characters.

Characters 32 to 127 are standard in all ASCII based systems and relate directly to the characters on your keyboard. ASCII numbers 1 to 31 are usually used for both control codes and various non-standard characters.

Characters 128 to 255 are not standard, and are often unique to your computer system or font, sometimes including international characters, mathematical symbols, graphic symbols, etc.

Note: DynaCADD can access these special characters through the Text Editor by pressing, in order, the % character twice, immediately followed by a three digit number, ie; %%233. For more details, including the character look up table, refer to Appendix H.

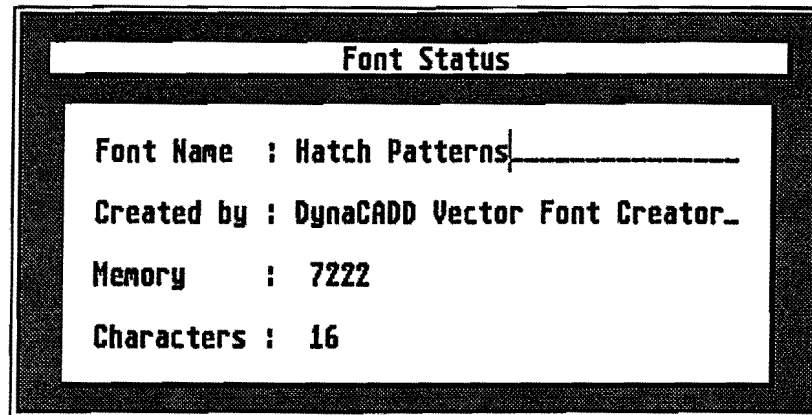


fig. (224) Font Status Dialog

Font Status

Each font can have its own name, aside from the filename under which it is saved. Select Font Status from the File menu to see or change the font name. This dialog also tells you the name of the font's creator, the amount of memory the font uses, and the total number of characters defined.

To change the font's name or the creator's name, use [Up arrow] or [Down arrow] to move the cursor to that line. Press [Esc] to clear the entry or use [Backspace] to erase single characters. Enter the new information, up to 29 characters, and press [Return].

The Drawing Window

All characters are created and modified in this area of the screen, one character at a time. Commands to create and edit characters and vectors are located either in the menus or as one of the command buttons, to the left of the drawing window.

There are several **boundary** (dashed) lines displayed in the drawing window, which indicate the horizontal and vertical limits of a character:

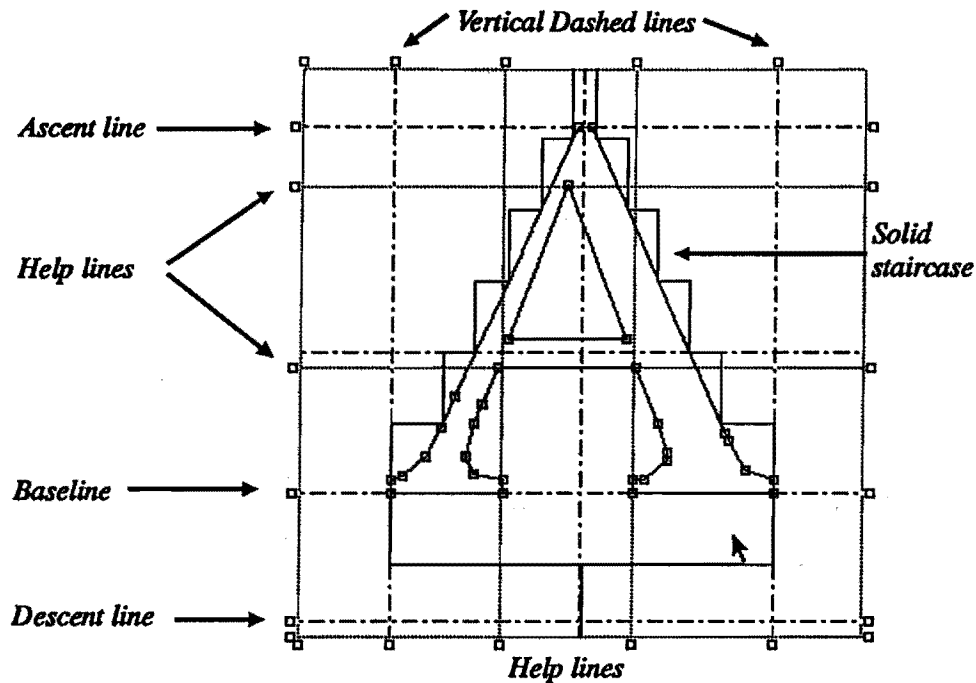


fig. (225) The Drawing Area

Baseline is the second horizontal line from the bottom. It is the line on which all uppercase letters (capitals) and any lowercase letters without descenders (ie; "a", "b", "f", "m", etc.) are placed.

Descent line represents the lower limit to which the descenders of such letters as "g" or "y" can extend below the baseline.

Ascent line, is the height to which the ascenders of characters such as "b", "d" and "f", may rise above the x-height. These two lines represent the lower and upper limits, respectively, all characters in a font.

Baseline, ascent and descent lines are the same for every character in a font. If you change one in any character, it is changed for every character.

Note: DynaCADD uses the position of these lines in the character “A” to determine text entity attributes and spacing.

To move any of these lines, drag the small box beside a line, on any side of the drawing window, and move it to the preferred location.

The vertical dashed lines are the left and right boundaries of the font, used by DynaCADD for correctly positioning characters in proportional spacing. They define the widest section of the character. These lines are unique for each character in a font. They, along with the vertical center line, are positioned automatically by the font editor when you click on Calc Kerning. You can also move the left and right borders manually, to define the vertical font borders, the same way you move the ascent and descent lines.

The solid “staircase” lines on both sides are the kerning guides. Kerning is a method of reducing the space between printed or displayed characters to bring them together for a more natural appearance and to reduce the whitespace between them. DynaCADD uses these guides to fit kerned characters together. Kerning is discussed further, below.

Helplines are user set guides for character construction. Initially, none appear in the drawing window, but they can be positioned by dragging one of the small boxes at the corners of the window to the desired position. You can have up to ten vertical and ten horizontal helplines. Helplines are global, that is, once set in one character, they remain in place for all characters in a font. They are saved with the characters when you save the font.

A character set can be measured vertically by its x-height, ascender and descender sizes. The x-height, or body size, is the vertical size of the lowercase letter “x”, measured from the baseline. This is the most important area of any letterform for 90% of the lowercase characters, since it conveys the impression of the size of a letter. There is no preset x-line in the drawing window, since it can be arbitrarily set for each font. Instead, to define an x-line, drag a horizontal helpline (see below) to the appropriate location.

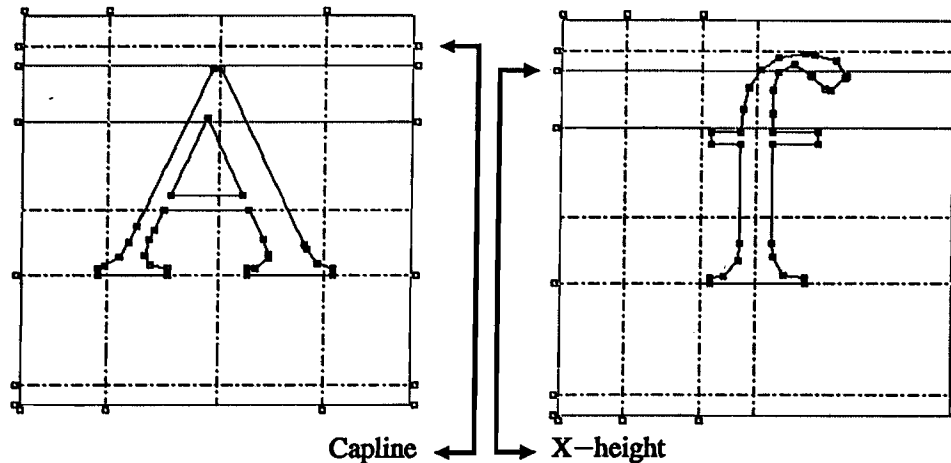


fig. (226) Characters with X-height and Capline Marked

The horizontal line not preset in the window is the capline. This line is usually spaced a little lower than the ascender line and marks the maximum height of capital letters. Ascenders normally extend a small amount above the height of uppercase characters. Again, this line has no predetermined size and should be set with a helpline.

The display of all lines in the drawing window can be turned on and off by selecting that type of line in the edit menu.

The X and Y coordinates of the pointer are displayed at the upper right corner of the screen. These change as you move the pointer on the drawing window. Position 0, 0 is located in the upper left corner of the drawing window. Also, when performing rotations, 0 degrees is to the right (east), not at the top of the drawing window.

The current character number (character slot) and the font name are also displayed, to the left of the drawing window.



fig. (227) Character Number and Font Name

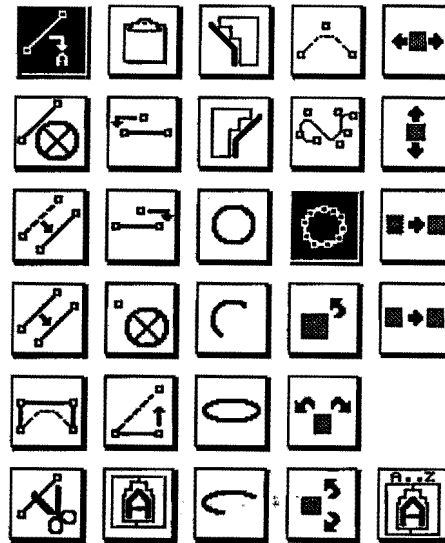


fig. (228) The Drawing Tool Panel

The Drawing Tools

Creating a character is done by drawing an outline in the drawing window. The process is similar to using the “rubber band” continuous line mode used in many paint or drawing programs.

There are several groups of related drawing tools in the font editor: those dealing with vectors, those dealing with control points, those dealing with shapes or curves and those dealing with entire characters. These groups are organized in vertical columns of buttons to the left of the drawing window. There are also related commands in some of the menus, including grid and snap, and zoom levels.

Undo is very important. It reverses the last action in the drawing window, unless a new tool was selected since the action was taken. In some cases, Undo also deletes the last vector or shape drawn.

Note: When selecting a vector for an operation, you must click on a control point, not a connecting line. The control points define the character, the lines simply connect these points.

View

Zoom In 2x	
Zoom In 4x	
Zoom Out 2x	
Zoom Out 4x	
Zoom Previous	
Zoom All	

Zoom / View

Often, when working with the tools, you need more detail. The font editor allows you to magnify the work area in the drawing window up to 32 times, in increments of two or four. All tools and functions work in every zoom level.

To magnify an area, select **Zoom In 2x** or **Zoom In 4x**. A box appears, and the pointer changes to a hand. This box represents the portion of the drawing area to magnify. Position the box and press the left mouse button. The area inside the box is magnified to fill the drawing area.

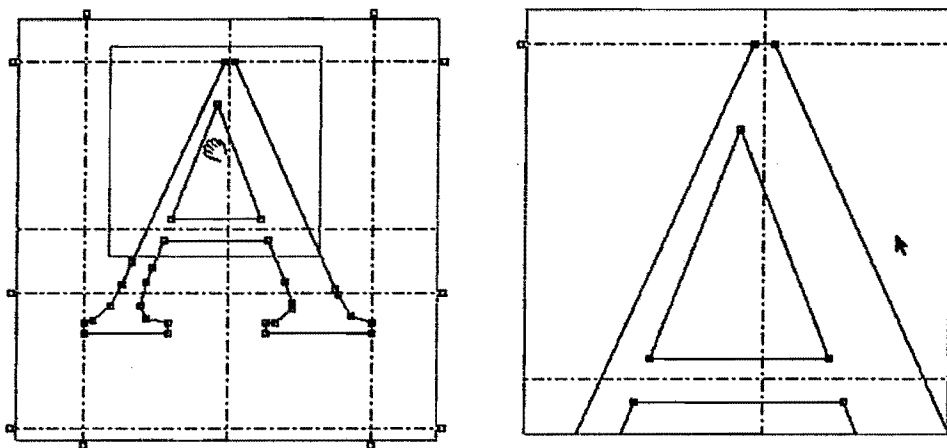


fig. (229) Zoom in on the Original Character

You can zoom in several times, increasing the magnification of an area until you reach the maximum zoom level. When you can use a zoom increment no further, the command appears grey in the menu.

The section of the drawing window under magnification can be changed by moving the sliders at the bottom and right sides of the window.

To reduce magnification one step, select **Zoom Out 2x** or **Zoom Out 4x**. **Zoom All** returns the drawing window to its original (normal) level.

Zoom Previous goes back to the last level of magnification.

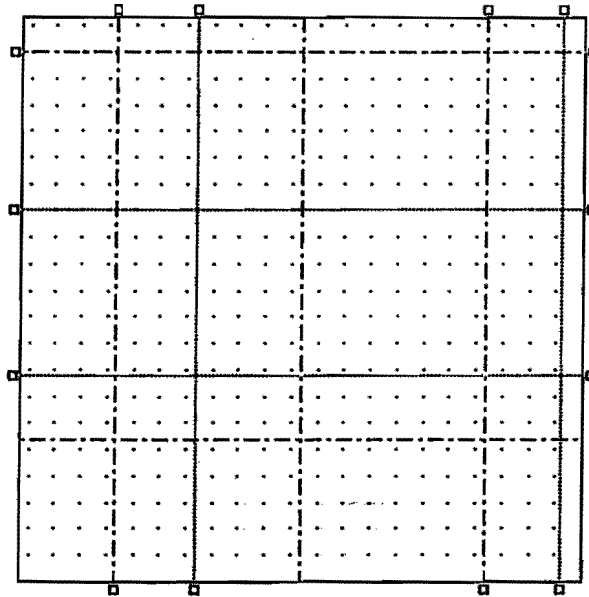


fig. (230) Drawing Area with Grid Displayed

Tools

Snap
Grid
Set Values

The Grid and Snap

The grid is simply an optical guide for designing characters. A dot appears at the intersection of points to indicate the grid location. The grid can be turned on and off by selecting the Grid option from the Tools menu.

Snap forces a newly placed or a moved control point to jump to the closest intersection that matches the snap intersection value. Existing control points not currently located on snap intersections are not affected until they are moved. If you move a vector with snap on, only the selected control point snaps into place, the others retain their relative positions to each other regardless of the snap. Snap is turned on and off by selecting the Snap option from the Tools menu.

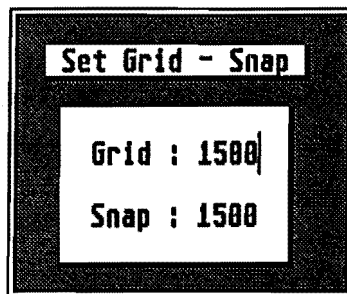


fig. (231) Set Grid and Snap Values

The size of the grid and snap intersection is set by selecting Set Values from the Tools menu. The grid and snap values always form a square, with X and Y values been equal distances. Press [Return] or click the pointer in the dialog to set the values. The minimum value for the snap and the grid is 500 points.

The grid is not displayed if the setting is too low for the current zoom level and the output would be too dense for use.

Grid and Snap sizes can be set to different values, although it makes more sense to make them the same, since snap intersections are not visible. However, on a normal monitor, you may find some grid values are simply too small to display properly at normal magnification. Since the drawing window is 64000 points on a side, a low grid value cannot be properly distinguished on the screen. This is why we permit different snap and grid values: you can set the snap value lower and use a suitably higher grid value, for example, a grid of 100 but a snap of 50.

Another suggestion for dealing with low grid values is to turn the grid off in low or normal magnification, and only display it in the higher zoom levels.

Vector Tools

A vector is a contiguous group of control points, joined by lines that define the character outline. The leftmost column beside the drawing window contains the vector tools. These tools affect entire vectors.

Note: Vectors cannot be joined together.



New Vector: Select to begin drawing a new vector. Move the pointer to the drawing window, where you want the vector to begin. Click the left mouse button. This sets the first control point. Move the pointer to where you want the next control point and click the mouse button and continue to add points. When you have all the control points established, press [Return].



Delete Vector: Click here, then on any control point of a vector you want deleted. Caution: deleted vectors cannot be restored. To copy or cut vectors, use the vector buffers.



Move Vector: Click here, then on any control point of a vector you want moved. A "shadow" of the vector moves with the pointer. Click the left mouse button to set the vector in the new location. To move only a section of the vector, use Move Cpoint (see below).



Copy Vector: Click here, then on any control point of a vector you want copied. Move the vector to the new location and press the left mouse button. This only makes one copy, unlike the paste function in the vector buffers.



Smooth Vector: Click here, then on any control point of a vector you want smoothed. This creates gradual curves between control points by adding extra points between existing

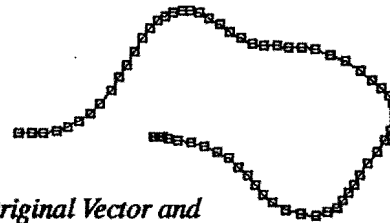
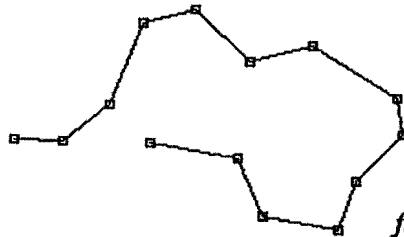
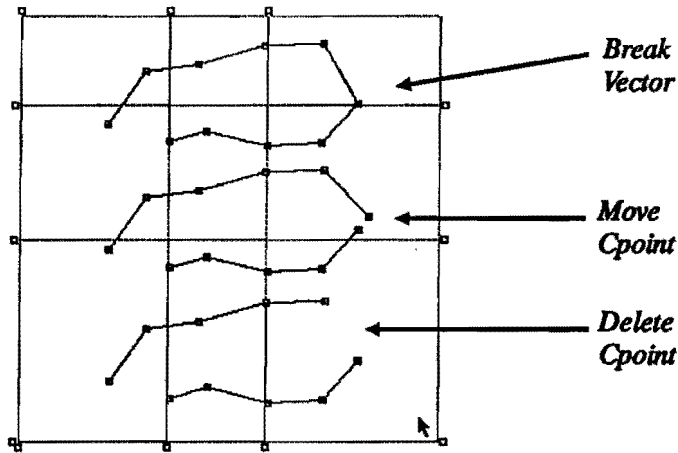


fig. (232) Original Vector and Smoothed Vector

fig. (233) Vector Modifications



Break Vector: This separates a vector into two or three parts by cutting it at the selected control points. Click here, then on the control point of a vector, where you want it broken. The vector is now broken into two separate vectors, at that control point. The selected control point is duplicated, to become the end points for the new vectors.

If you want to break a vector into three sections, click on a first control point, then click a second time on another control point of the same vector to define the end of the middle section. The selected control points are duplicated, to become the endpoints for three new vectors.

If you want to shorten a line, use Delete Cpoint.



Vector Buffers: Described above. These store selected vectors in ten buffers that can be saved to or loaded from disk. Note that vector buffers are not saved to disk when a font is saved: they must be saved separately.

To close a vector, that is: make the final point join with the first, press [c] before you press [Return].

Control Point Tools

These tools affect only a single control point (cpoint) at a time. The display of control points can be turned off and on by selecting that option from the Edit menu. A single vector can contain as many as 127 control points. A character can contain a very large number of control points.



Add Cpoint Before: Click here and then on the control point before which you want to add another control point. Before depends on the direction in which the vector was created. For example, if you created it from left to right, the control point appears left of the point you select.



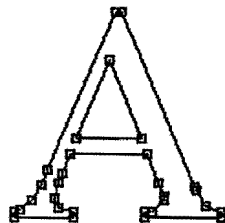
Add Cpoint After: Click here and then on the control point after which you want to add another control point. This command depends on the direction in which the vector was created. For example, if you created it from left to right, the control point appears right of the point you select.



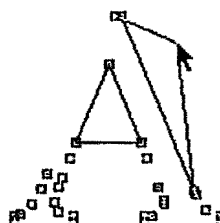
Delete Cpoint: Click here then on the control point you want to delete. A deleted control point can be restored by selecting Undo from the Edit menu.



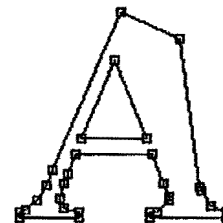
Move Cpoint: Click here then on the control point you want to move. A moved control point can be restored by selecting Undo from the Edit menu.



Original Character



Move Cpoint



Result

fig. (234) Control Point Manipulation

Kerning Tools

Kerning, as described earlier, affects the relationship between adjacent characters. The kerning guides are rectangles which enclose the outer edges of a character and tell DynaCADD where characters can overlap. Traditionally, kerning was done only on selected pairs of characters, for example Yo, We and WA. Depending on the limits of the equipment, this could be a list of between 20 and 200 pairs. But with DynaCADD, every character can be correctly kerned with every other character, according to the kerning guides.

Although you normally let the font editor calculate the position of the kerning guides automatically by clicking Kern All, you also have the option of setting the guides yourself. Whichever method you select, you must set the kerning guides for each character separately.

Unlike most desktop publishing systems, letter kerning is done automatically in DynaCADD, based on the information provided by these guides. You cannot manually adjust kerning values in DynaCADD, although you can adjust individual character spacing.



Calc Kerning:

Select this to calculate the kerning guides for both sides of the current character automatically. This must be done for each character separately. Automatic kerning also adjusts the right and left boundary guides for that character.



Right Kerning:

Lets you manually set the right hand kerning guides of the current character by dragging the various sections to new locations. It does not affect the right boundary lines.



Left Kerning:

Lets you manually set the left hand kerning guides by dragging the various sections to new locations. It does not affect the left boundary lines.



Kern All:

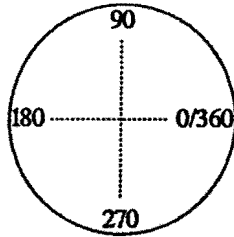
Click her to automatically kern the complete character set.

A V

fig. (235) Kerning, Before and After

AV

Shape and Curve Tools



The font editor features several basic shapes and curves to make character and symbol design easier. The smoothness of their curves depends on whether the High Control is selected.

Although the circle and ellipse appear as continuous objects, they have definite start and end points, at the zero (0) degree mark. This is the rightmost position in the drawing window.

Shapes and curves are vectors and can be changed, broken and deleted like any other vector.



Circle:

Select to draw a circle. Click the pointer at the position where you want the center of the circle to be, then move the pointer. The outline of the circle follows. Click the left mouse button to generate the circle when the appropriate size is reached.



Arc:

Select to draw an arc (a portion of a circle). Click the pointer at the position where you want the center of the circle to be, then move the pointer. The outline of the circle follows. Move the pointer to the location where you want the arc to begin. Click the left mouse button to establish the circle's diameter. Move the pointer again: the arc grows or shrinks, according to the direction you move the pointer. Click the left mouse button to generate the arc.



Ellipse:

Select to draw an ellipse. Click the pointer at the position where you want the center of the ellipse to be, then move the pointer. The outline of a circle follows. This determines the horizontal axis of the ellipse. Click the left mouse button to establish the axis. Move the pointer again: the ellipse grows or shrinks vertically, according to the direction you move the pointer. Click the left mouse button to establish the vertical axis and generate the ellipse.



**Elliptical
Arc:**

Select to draw an elliptical arc (portion of an ellipse). Click the pointer at the position where you want the center of the ellipse to be, then move the pointer. The outline of a circle follows. This determines the horizontal axis of the ellipse. Click the left mouse button to establish the axis. Move the pointer again: the ellipse grows or shrinks vertically, according to the direction you move the pointer. Click the left mouse button to establish the vertical axis: a line joins the pointer and the central point. Move this to the place where you want the arc to begin. Click the button. Now move the pointer to define the arc. Click the mouse button to generate the arc.



Bézier:

This creates a cubic Bézier curve, based on three control points. Click the pointer in the drawing area to set the first control point, then again in different locations to set the next two. The curve is automatically fitted to the points when you press the button.

The last point of the curve becomes the first point of the next set of curves, so you can continue creating contiguous Bézier curves until you select another tool. Each curve is, however, a separate vector.



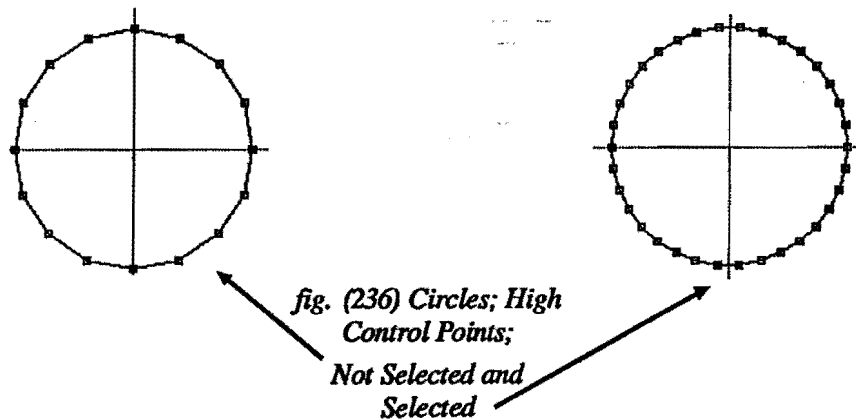
B-spline:

This creates B-spline curves, which are similar to Bézier curves, except that you can add any number of control points (not merely three) and the curve fit to the control points is better than Bézier curves. Create a vector exactly as you would a new vector, by clicking for every desired control point. When you press [Return], the B-spline curve is fitted to those points.

**High Control
Select:**

This determines the number of control points generated when smoothing a vector or creating any of the shape and curve vectors. When highlighted, more control points are inserted, resulting in a smoother curve. If this generates more control points than you want, turn it off and try creating the vector without it.

To create irregular polygons, press [c] before you press [Return]. This closes the vector by joining the startpoint and endpoint with a line.



Character Tools

These tools affect the entire character: every vector and control point visible in the drawing window. Individual elements can be manipulated with any of the previously described commands.



Rotate: This rotates the character any number of degrees, around the center of the window. A circle appears at the center of the character and a line denoting the angle appears attached to the pointer. Move the angle marker to the correct location and press the button. The character is rotated.

Note: The zero degree position is at the far right, not at the top of the window as in maps.



Flip-X: Click here to flip the character left to right. Click again to reverse the flip.



Flip-Y: Click here to flip the character top to bottom. Click again to reverse the flip.

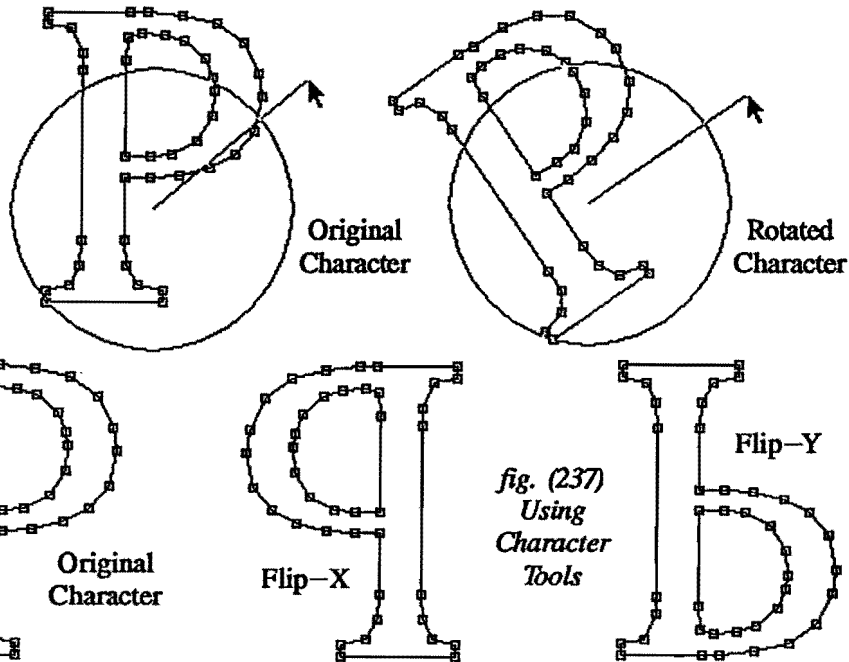
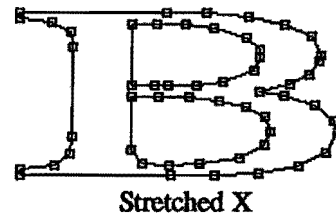
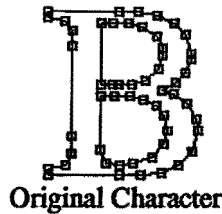


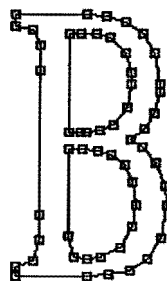
fig. (238)
Character
Stretching



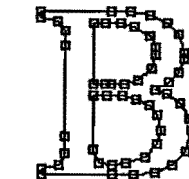
Stretch X: This stretches or shrinks the character along the X axis. It works in mirror mode only, away from or towards the vertical center of the character. That is, if you stretch (or shrink) a control point on the left side of a character further left, the entire left side of the character moves to the left by the same ratio as the distance indicated. The right half is also stretched further right an equal amount. To activate, click on any control point. A line appears denoting the distance to stretch. Vertical (up and down) distance is ignored; only the horizontal distance is relevant. Click the mouse to stretch the character.



Stretch Y: This stretches or shrinks the character along the Y axis. It works in mirror mode only, away from or towards the horizontal center of the character. That is, if you stretch (or shrink) a control point at the top of a character further towards the top, the entire top half of the character is stretched by that amount. The bottom half is also stretched further down by an equal amount. To activate, click on any control point. A line appears denoting the distance to stretch. Horizontal (left and right) distance is ignored; only the vertical distance is relevant. Click the mouse to stretch the character.



Because the stretch commands manipulate the entire character around an axis, you should not drag the stretch indicator across the relevant mirror axis (vertical for X, horizontal for Y). Also, if your character is entirely in one half of the window and does not cross the axis, then stretching in the direction of the axis generates unexpected (and often unwanted) results.





Move:

This moves the entire character in the direction and the distance indicated. Click anywhere in the window to indicate a start point. Then drag the move indicator any distance and angle. When you click the button again, the character moves that far, along the path determined by the angle. Movement is determined from the center of the window. If you click on a control point, then the character move is determined from that control point, rather than the center of the window. This is a more precise method of moving a character.

When a character collides with the borders of the window, using stretch or move, control points are not lost. They are, instead, forced into a straight line parallel with the edge of the window. Unless this is what you want, select Undo and try again, but use either a smaller distance or select a specific control point to move.



Copy:

This command is functionally identical to the move command (above). The only difference is how the original vectors are treated. With the move command the original vectors are shifted. The copy command first produces an image of the character and then moves the image the specified distance.

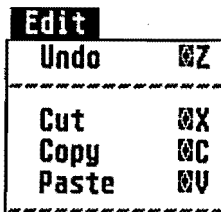
Cut, Copy:

These options are found in the Edit menu. They are the same as vector cut and copy, except that they work on the entire character. Cut removes the character from the window into the buffer. This is a simple method for clearing a window completely. The character remains in the buffer until another cut or copy overwrites it. Copy is the same as cut, except that the character remains in the window.

Paste:

This is somewhat different than vector paste. It pastes the contents of the cut/copy buffer, but performs more like the move command (above). Click once, anywhere in the window. This sets the start point for the paste. Drag the move indicator any distance and angle. When you click the button again, the character is pasted that far away from the original location, along the path determined by the angle.

If you paste a copied or cut character into a blank character slot, there is no movement required. The copy is located at the same coordinates as the original.



File Commands

These commands, in the File menu, are for loading and saving font files or vector buffers. All font files (.FNT) and vector buffer files (.BUF) are normally stored in the FONTS directory or folder.

Open Font: Load a font. This overwrites any font currently in memory.

Save Font: Saves the current font under the existing name. If no name was used, you are presented with the file selector and must enter a name and extension (.FNT).

Save Font As: Saves the current font under a new name. You are presented with the file selector and must enter a name and extension (.FNT).

Load Buffers: This loads a set of ten vector buffers and overwrites any buffers currently in memory.

Save Buffers: Saves the current vector buffers. You are presented with the file selector and must enter a name and extension (.BUF).

Select Character:

File	
Open Font...	⌘O
Save Font	⌘S
Save Font As...	

Load Buffers	
Save Buffers	

Select Character	
Font Status	

Quit	⌘Q

Use this command to select a character to display, create or edit. You are presented with the existing system character set as a reference. Your choice does not change the system set. The character slot currently on screen on the drawing window is highlighted. If a slot contains a character on the font memory, it is shadowed. To see other available characters, click on page left or page right. To select a character, click on its button, then click OK.

When designing a character, you can also use the [+] and [-] keys to display other characters in the set.

Font Status: This displays the font name, creator, the amount of memory used and the number of characters used. See above for more information.

Quit: Select this to exit the program and return to the desktop. Make sure you save your work before you quit.

Hatch Patterns

Hatch patterns are contained in the font file HATCH.FNT. These are used for the INSERT HATCH command (see Chapter 7). HATCH.FNT must be in the FONT folder, otherwise hatch patterns are disabled in the Set menu.

You can modify the existing patterns or add your own in the empty character slots. New patterns appear in the hatch pattern dialog in DynaCADD.

Hatch patterns do not require kerning.

Hatch patterns should be symmetrical in design in order to be properly displayed when inserted into a drawing. Vectors should be evenly spaced between the borders and form regular groups.

Lines (vectors) should align to meet as if they wrapped around right to left and top to bottom. Diagonal vectors should also wrap across the borders, right to left. This is easily seen if you load HATCH.FNT and examine the existing patterns.

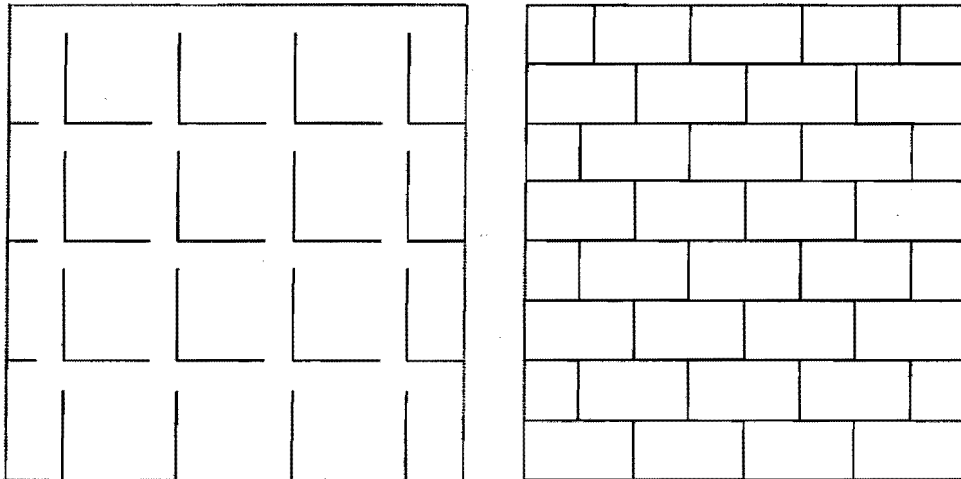


fig. (239) Hatch Patterns Examples

If your hatch pattern is composed of small groups of vectors, such as those used in terrain symbols, the groups should align both horizontally and vertically and be evenly spaced, not randomly created and spaced.

Creating a hatch pattern is easy, but may require some experimentation to get the pattern to align properly and symmetrically when inserted into a drawing. It is sometimes easier to use an existing pattern as a template, to determine vector placement for wrap around, then delete unnecessary vectors when finished.

Design Suggestions

1. Plan to design a full character set, not simply uppercase characters. All uppercase is more difficult to read and offers a less visually interesting appearance.
2. The visual aspect of characters is more important than the mathematical aspect. Design your characters for visual appeal, rather than trying to make them conform to exact measurements. For example, rounded letters like "O", "C" and "G" look smaller than other letters unless drawn slightly larger.
3. Rounded capital letters such as "C", "G" and "O" are usually designed to be slightly taller than straight letters such as "I", "T" and "L", so they are placed slightly below the baseline.
4. Because of the thin size of certain characters, you may want to increase the kerning space around such characters as "I", "i", "l" and "I", in order to improve their appearance when kerned with other wider characters.
5. Serifs are small lines that extend, usually horizontally, from the free ends of letter strokes and provide a visual guideline when reading. Serif typefaces (such as Baskerville, Bodoni, Times and Garamond) are easier to read in large bodies of text than sans serif fonts (such as Helvetica, Futura, Franklin and Univers). Sans serif fonts should be reserved for headlines, titles, or single text lines. Serifs themselves can be any of several shapes and sizes, including slab (square), wedge, hairline and bracketed (cupped).
6. Simple fonts are easier to read than decorative, script, or the various display fonts.

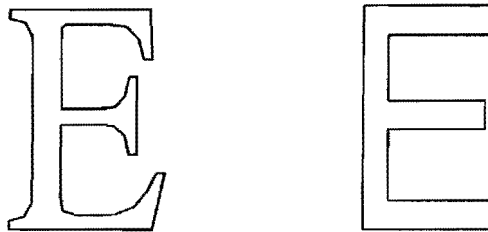


fig. (240) Serif and Sans Serif Characters



fig. (241) Aligned and Non Aligned Numbers

7. Remember that vector fonts are outline or line fonts; they are not filled or solid fonts.
8. Numbers can be designed in two styles: aligning and non aligning. Aligning numbers are all the same size and align along the baseline. Non aligning numbers are small, with ascenders and descenders.
9. Italic fonts have characters slanted to the right, usually 10 or 12 degrees from the vertical. These are harder to read in large bodies of text than unslanted fonts, so use italics for emphasis and short pieces of text. DynaCADD can slant text within the program. Although this is not pure italics in a typographic sense, it is sufficient for most uses, so there is no need to design a separate italic character set.
10. The ratio of space between descent baseline, X-line baseline and ascent baseline is not fixed. Use the visual appearance of a font to guide you. You should purchase one of the many typeface guides available, published either as books or as commercial type catalogues from such companies as Compugraphic, and examine the many samples they offer. Use a pica point ruler, available from any art supply shop, to measure the various fonts.
11. You can also use vectors as guides for letter design, especially circles, ellipses and arcs. Copy them into the vector buffers and save them to disk for use later.
12. It is unlikely that you will use all 255 characters for a font. However, it is always a good idea to designate at least one character as a blank (for example, number 1 or number 255). You can use this blank as a temporary canvas on which to work, experiment, make adjustments to single vectors and so on.

13. Undo is a valuable function. It restores any change, deletes vector, shape or curve, unless you have switched to another tool. However, Undo only restores the last change.
14. Make sure your kerning guides have been set for the current character, either manually or automatically, before continuing on to another character.
15. Save your font after each character you have created or changed.

Suggested Reading:

Designing With Type: A Basic Course in Typography, by James Craig, Watson-Guptill Publications, New York, NY, 1980. A good general book on type and how to use it. Many useful examples.

The Thames and Hudson Manual of Typography, by Ruari McLean, Thames and Hudson, London, UK, 1980. A good overview of the craft, including its history, modern developments and design considerations.

The TypEncyclopedia: A User's Guide to Better Typography, by Frank J Romano, R.R Bowker Company, New York, NY, 1984. An easy to read encyclopedia about typography.

Notes on Graphic Design and Visual Communication, by Gregg Berryman, William Kaufman Inc., Los Altos, CA, 1984. A general approach to design, with pages on using type.

Rookledge's International Type Finder, by Christopher Perfect & Gordon Rookledge, PBC International, New York, NY, 1983. A guide to typefaces and how to recognize them, with many samples and charts.

The Chicago Manual of Style, 13th Edition Revised and Expanded, The University of Chicago Press, Chicago, USA, 1982. Not only excellent material on type and design, but on style, usage and grammar as well.

Font Editor Error Messages

**Unable to load resource file FONTEDIT.RSC.
Please install and retry.**

All DynaCADD resource files should be in the same folder or directory. Copy the font editor resource file to the RESOURCE directory, then load the program again.

**Unable to load Font file FONTEDIT.FNT.
Please install and retry.**

The default font file must be kept in the FONT directory. Copy the file to the correct directory and load the editor again.

**Too few control points. Unable to break vector.
Please reselect vector.**

There are not enough control points to break the selected vector apart. A break requires at least three control points (including endpoints). Either select another vector to break or add extra control points to this one, before trying again.

**Unable to break vector at the endpoint.
Please reselect vector.**

You cannot select an endpoint as a location to break a vector. You must select control points between two endpoints. Try again.

Too few control points. Unable to delete cpoint.

There are not enough control points on this vector to remove one. Select another vector.

Too many control points. Unable to add cpoint.

The selected vector already has 127 control points, the maximum allowed on a single vector. You cannot add any more to it. To add points, you must first break the vector into two or more parts.

The selected filename exists. Do you wish to replace it?

You are trying to save your font or vector buffers under a name that already exists for another file. If you replace the existing file, everything in it is overwritten by what is in memory. You can cancel and select another name for your file.

A file error has occurred while attempting to save the current font.

This is a general purpose error message which results from any of a number of problems. Your disk might be write protected, or the file might be set as write only. Check your disk before continuing. You might also try saving the file with a different name.

A file error has occurred while attempting to open the selected font.

This is a general purpose error message which results from any of a number of problems. It usually happens when you enter a name for a file that does not exist in the directory, or on the disk. Try selecting a name from the file list.

A file error has occurred while attempting to save the vector buffers.

This is a general purpose error message which results from any of a number of problems. Your disk might be write protected, or the file might be set as write only. Check your disk before continuing. You might also try saving the file with a different name.

Quitting without saving will lose any information in the current Font that has not been saved.

You have made changes to the font in memory but have not saved it. If you quit now, all changes are lost. You can continue or cancel and save the file, before quitting.

Unable to load FONTEDIT.ALT

The error message file must be kept in the CONFIG directory. Copy the file to the correct directory and load the editor again.

File Transfers ... Appendix A

Drawing Interchange Files (DXF)

Drawing Interchange Files are the industry standard for transferring drawings between various microcomputer CADD packages. Virtually all major CADD packages can read and write DXF files. DXF files can be recognized by their ".DXF" extension.

DXF Conversion Process

DynaCADD supports the following DXF entity types:

- points
- lines
- arcs
- circles
- text
- polylines
- traces
- solids
- faces
- blocks

As DynaCADD distinguishes between 2D and 3D modes, be sure to select the appropriate mode during DXF import and export.

Block definitions from the DXF file are temporarily stored on the current disk as DynaCADD compatible figures. Make sure that you have sufficient disk space when importing large DXF files.

Named layers and colors are retained and stored in the order in which they appear in the DXF file. GCP to UCS, UCS to GCP translation is supported.

When you import a DXF file, all text in the file is set to the Leroy Rough font (LEROY_R.FNT). This font must be in your FONTS folder. It is loaded automatically when you import a DXF file.

When you export a DXF file, font information is not included, but the character height and size information is retained. The fonts in the DynaCADD drawing are all changed to the DXF 'standard' font in the export file. Speciality fonts or character sets in DynaCADD are not retained in the DXF file.

Appendix A, File Transfers

All currently active entities are converted to DXF format. Entities that have been Masked are not converted.

Note: Use DEF to transfer files between DynaCADD running on different systems.

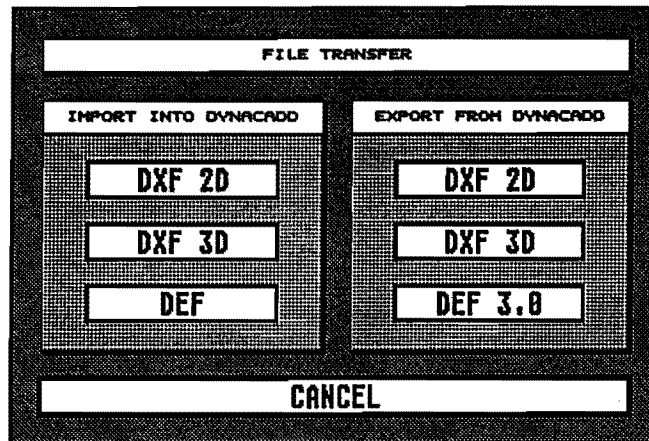


fig. (242) File Transfer Dialog

DynaCADD Exchange Format (DEF)

DynaCADD Exchange Format capabilities are included in all releases of DynaCADD. DEF files insure transfer of drawings between DynaCADD on various computer systems and upward compatibility to new versions of DynaCADD. DEF files can be recognized by their ".DEF" extension

DynaCADD writes all 2D and 3D entities, including masked entities, to a DEF file. Drawing information, as well as these parameters are stored in the DEF file:

- Views • Images • GCPs
- Units • Sizes • Scales

DEF import and export can be canceled at any time by clicking the right mouse button.

DEF 3.0 File Format Specification

A complete file consists of the following sections:

- Header
- Layer Table
- Style Table
- View Table
- GCP Table
- Image Table
- Color Table
- Floating Global Variables
- Entity Definitions

The following descriptions will be much clearer if you print a small DEF file in DynaCADD to follow as an example.

Appendix A, File Transfers

Header

Version	The Major version number multiplied by 100 plus the minor version number. For example the current DEF version number written by DynaCADD 2.04 is 300. This is the one header variable that is not optional. ALL DEF 3.0 files must include the Version number 300.
Scale	A floating point value representing the drawing scale. For example, 0.5 represents one-half scale.
Units	An integer index to the selected unit base: 1 = inches 2 = feet 3 = yards 4 = miles 5 = millimeters 6 = centimeters 7 = meters 8 = kilometers
Width	The width of the drawing specified in the currently selected units.
Height	The height of the drawing specified in the currently selected units.
Line__Scale	The global line style scale.
Origin__X	The 2D drawing origin along the X axis.
Origin__Y	The 2D drawing origin along the Y axis.
Color__By__Layer	A flag determining the state of the Color__By__Layer system variable: Off = 0 On = 1
Style__By__Layer	A flag determining the state of the Style__By__Layer system variable: Off = 0 On = 1

Appendix A, File Transfers

Weight__By__Layer A flag determining the state of the
Weight__By__Layer system variable:
Off = 0
On = 1

Comments Any line preceded by *** is considered to be a
comment and is ignored.
*** Sample DEF 3.0 Header

Version
300
Scale
1.0
Units
1
Width
17.0
Height
11.0
Line__Scale
1.0
Origin__X
0.000000
Origin__Y
0.000000
Color__By__Layer
0
Style__By__Layer
0
Weight__By__Layer
0

*** End of Sample Header

Layer Table

The layer table is optional and is used to define all layer parameters. A layer definition consists of the following structure:

```
Layer__Definition
layer index          (integer 0..255)
layer name           (string)
layer lock status    (integer)
                     1 = locked
                     0 = unlocked
layer visibility      (integer)
                     1 = visible
                     0 = invisible
layer color           (integer 1..15)
layer style           (integer 0..63)
layer weight          (integer 0..2)
End
```

*** Sample Layer Definition

```
Layer__Definition
0
SAMPLE__LAYER
0
1
1
0
1
End
```

*** End of Sample Layer Definition

The preceding sample definition names layer 0 as SAMPLE__LAYER, and declares it to be unlocked and visible. The layer color is 1, using style 0 from the style table and the layer weight is set to 1.

Style Table

The style table is optional and is used to define all line styles. A style definition consists of the following structure:

Style__Definition	
style index	(integer 0..63)
style name	(string)
style local scale	(floating point)
style string	(string)
End	

*** Sample Style Definition

```

Style__Definition
0
SAMPLE__STYLE
1.50
+++++ +++++ +++++ +++++
End

```

*** End of Sample Style Definition

The preceding sample definition names style 0 as SAMPLE__STYLE, with a local scale of 1.50. The style string is the line pattern, constructed according to the manner explained in Chapter 5.

GCP Table

The GCP table is optional and is used to define all Geometric Coordinate Planes.

A GCP definition consists of the following structure:

GCP__Definition	
GCP index	(integer 0..15)
GCP name	(string)
plane rotation about the Z axis	(angle - degrees)
plane rotation about the X axis	(angle - degrees)
plane rotation about the Y axis	(angle - degrees)
GCP X Translation	(floating point)
GCP Y Translation	(floating point)
GCP Z Translation	(floating point)
End	

*** Sample GCP Definition

```
GCP__Definition
0
Top
0.000000
0.000000
0.000000
1.000000
2.000000
0.000000
End
```

*** End of Sample GCP Definition

The preceding sample definition names GCP 0 as Top with no rotation from the absolute 3D plane, and an X–Y–Z translation of 1.0, 2.0, 0.0 from the absolute 3D origin.

View Table

The View table is optional and is used to define all active 3D views.

A View definition consists of the following structure:

```

View__Definition
View index                (integer 0..3)
View GCP index            (integer 0..15)
X location of lower left corner of View bounds (integer)
Y location of lower left corner of View bounds (integer)
X location of upper right corner of View bounds (integer)
Y location of upper right corner of View bounds (integer)
X location of View origin (integer)
Y location of View origin (integer)
View Zoom factor          (floating point)
plane rotation about the Z axis (angle - degrees)
plane rotation about the X axis (angle - degrees)
plane rotation about the Y axis (angle - degrees)
View Scale factor         (floating point)
Scroll distance of view origin along the 2D X axis (floating point)
Scroll distance of view origin along the 2D Y axis (floating point)
End
  
```

*** Sample View Definition

```

View__Definition
0
1
0.000000
0.000000
34.000000
22.000000
17.000000
11.000000
1.000000
  
```

Appendix A, File Transfers

0.000000
0.000000
0.000000
0.250000
0.000000
0.000000
End

***** End of Sample View Definition**

The preceding sample defines view 0 as having GCP 1 as it's prime rotation viewing plane. The bounding area of the view is encompassed by the coordinates 0.0, 0.0 to 34.0, 22.0 with the origin in the center at coordinate 17.0, 11.0.

The view zoom factor is 1.0. The current viewing plane has not been rotated from the absolute 3D plane. The scale factor is set to 0.25 and the view has not been scrolled along the 2D X or Y axis.

Color Table

The Color table is optional and is used to define the full drawing color palette. A Color definition consists of the following structure:

```
Color__Definition
Color index                (integer 0..15)
Red Component of RGB color (integer 0..1000)
Green Component of RGB color (integer 0..1000)
Blue Component of RGB color (integer 0..1000)
End
```

*** Sample Color Definition

```
Color__Definition
0
800
800
800
End
```

*** End of Sample Color Definition

The preceding sample defines color 0 as having RGB components of R: 800 G: 800 B: 800 (Light Gray).

Note: Color indices 0 and 1 are used by the DynaCADD User Interface and should normally be left undefined in a DEF file. If they are defined they should be set to the following RGB values:

```
Color 0  R: 1000  G: 1000  B: 1000
Color 1  R: 0     G: 0     B: 0
```

Image Table

The Image table is optional and is used to define all Images. An Image definition consists of the following structure:

```
Image__Definition
Image index          (integer 0..15)
Image name           (string)
X location of lower left corner of Image (floating point)
Y location of lower left corner of Image (floating point)
X location of upper right corner of Image (floating point)
Y location of upper right corner of Image (floating point)
End
```

*** Sample Image Definition

```
Image__Definition
0
TestImg
2.500000
3.000000
5.000000
8.000000
End
```

*** End of Sample Image Definition

The preceding sample definition names Image 0 as TestImg with a bounding area encompassed by the coordinates 2.5, 3.0 and 5.0, 8.0.

Floating Global Variables

There are nine floating global variables used when defining the attributes of any entity definitions that follow in the DEF file. The variables are:

Layer
Color
Weight
Style
Group
Groupend
Mask
Unmask
Viewmask

Note: Floating variables may be placed anywhere in a DEF file with two restrictions:

- 1) They must never be placed in a Table.
- 2) They must never be placed where an ASCII name is expected.

Layer

Layer defines the current layer index assigned to all following entities. The range of valid layer indexes is 0 to 255.

```
*** Example
Layer
12
*** End of example
```

The preceding example sets the current layer index to 12.

Appendix A, File Transfers

Color

Color defines the current color index assigned to all following entities. The range of valid color indexes is 1 to 15.

```
*** Example
Color
3
*** End of example
```

The preceding example sets the current color to 3.

Weight

Weight defines the current weight index assigned to all following entities. The range of valid weight indexes is 0 to 2.

```
*** Example
Weight
1
*** End of example
```

The preceding example sets the current pen weight to 1.

Style

Style sets the current style index assigned to all following entities. The range of valid style indexes is 0 to 63.

```
*** Example
Style
2
*** End of example
```

The preceding example set the current style index to 2.

Group and Groupend

All entities that are grouped together as a DynaCADD figure should be preceded by the global name Group and end with the global name Groupend. These globals must not be nested. That is no Group label should appear between another set of Group and Groupend labels.

Note: Hatch patterns and section lines are treated as groups.

*** Example of grouping entities

Group

*** Entity definitions go here

Groupend

*** End of example

Mask and Unmask

All entities defined after a Mask command will not be displayed by DynaCADD. Unmask is used to reset the state of the mask flag.

*** Example of masking entities

Mask

*** Entity definitions go here

Unmask

***End of example

Viewmask

Viewmask is a bitwise flag that controls the generation of 3D entities in each active 3D View.

1 = View #1
2 = View #2
4 = View #3
8 = View #4

To mask a 3D entity in view #1 and view #3, Viewmask would be set to 5 (1+4).

```
*** Example
Viewmask
7
*** End of example
```

The preceding example, sets all 3D entities that follow to display in view #4, only. To reset the display to all views set Viewmask to 0.

Entity Definitions

The following list contains the specification for currently supported 2D and 3D entities. If entity types are added to DynaCADD, this list will be updated. You should therefore read entities in a table driven format and skip all unsupported entities.

All angles are given in degrees with the zero degree mark towards the east and the positive direction increasing in a counter-clockwise direction. All entity definitions must start with the entity name, as shown here, and finish with End.

2D Entities

2D__Point

X location	(floating point)
Y location	(floating point)
End	

2D__Line

X location of start	(floating point)
Y location of start	(floating point)
X location of end	(floating point)
Y location of end	(floating point)
End	

2D__Circle

X location of center	(floating point)
Y location of center	(floating point)
radius	(floating point)
End	

2D__Arc

X location of center	(floating point)
Y location of center	(floating point)
radius	(floating point)
start angle	(degrees - floating point)
end angle	(degrees - floating point)
End	

Appendix A, File Transfers

2D__Ellipse	
X location of center	(floating point)
Y location of center	(floating point)
major axis	(floating point)
minor axis	(floating point)
rotation	(degrees - floating point)
End	
2D__Ell__Arc	
X location of center	(floating point)
Y location of center	(floating point)
major axis	(floating point)
minor axis	(floating point)
rotation	(degrees - floating point)
start angle	(degrees - floating point)
end angle	(degrees - floating point)
End	
2D__Text	
X location, lower left corner of text string	(floating point)
Y location, lower left corner of text string	(floating point)
character width	(floating point)
character height	(floating point)
character slant	(degrees - floating point)
character rotation	(degrees - floating point)
text spacing type	(integer)
	0 - constant (mono spacing)
	1 - proportional
	2 - kerned
additional spacing	(floating point)
name of font (i.e., LEROY__R)	(string)
text string	(string)
End	

2D__B spline

X location of point #1	(floating point)
Y location of point #1	(floating point)
X location of point #2	(floating point)
Y location of point #2 (to 49 control points)	(floating point)
X location of last point	(floating point)
Y location of last point	(floating point)
End	

2D__Bezier

X location of point #1	(floating point)
Y location of point #1	(floating point)
X location of point #2	(floating point)
Y location of point #2 (to 12 control points)	(floating point)
X location of last point	(floating point)
Y location of last point	(floating point)
End	

2D__Solid

X location of point #1	(floating point)
Y location of point #1	(floating point)
X location of point #2	(floating point)
Y location of point #2	(floating point)
X location of point #3	(floating point)
Y location of point #3	(floating point)
X location of point #4	(floating point)
Y location of point #4	(floating point)
End	

3D Entities

All 3D coordinates are given in absolute 3D space and are NOT considered relative to the current GCP.

3D__Point

X location	(floating point)
Y location	(floating point)
Z location	(floating point)
End	

Appendix A, File Transfers

3D_Line

X location of start	(floating point)
Y location of start	(floating point)
Z location of start	(floating point)
X location of end	(floating point)
Y location of end	(floating point)
Z location of end	(floating point)
End	

3D_Circle

X location of center	(floating point)
Y location of center	(floating point)
Z location of center radius	(floating point)
plane rotation about the Z axis	(degrees - floating point)
plane rotation about the X axis	(degrees - floating point)
plane rotation about the Y axis	(degrees - floating point)
End	

3D_Arc

X location of center	(floating point)
Y location of center	(floating point)
Z location of center radius	(floating point)
start angle	(degrees - floating point)
end angle	(degrees - floating point)
plane rotation about the Z axis	(degrees - floating point)
plane rotation about the X axis	(degrees - floating point)
plane rotation about the Y axis	(degrees - floating point)
End	

3D_Ellipse

X location of center	(floating point)
Y location of center	(floating point)
Z location of center major axis	(floating point)
minor axis	(floating point)
rotation	(degrees - floating point)
plane rotation about the Z axis	(degrees - floating point)
plane rotation about the X axis	(degrees - floating point)
plane rotation about the Y axis	(degrees - floating point)
End	

3D__Ell__Arc	
X location of center	(floating point)
Y location of center	(floating point)
Z location of center major axis	(floating point)
minor axis	(floating point)
rotation	(degrees - floating point)
start angle	(degrees - floating point)
end angle	(degrees - floating point)
plane rotation about the Z axis	(degrees - floating point)
plane rotation about the X axis	(degrees - floating point)
plane rotation about the Y axis	(degrees - floating point)
End	
3D__Text	
X location, lower left corner of text string	(floating point)
Y location, lower left corner of text string	(floating point)
Z location, lower left corner of text string	(floating point)
character width	(floating point)
character height	(floating point)
character slant	(degrees - floating point)
text rotation	(degrees - floating point)
plane rotation about the Z axis	(degrees - floating point)
plane rotation about the X axis	(degrees - floating point)
plane rotation about the Y axis	(degrees - floating point)
text spacing type	(integer)
	0 - constant (mono spacing)
	1 - proportional
	2 - kerned
additional spacing	(floating point)
name of font (ie; LEROY__R)	(string)
text string	(string)
End	

Appendix A, File Transfers

3D__Bspline

X location of point #1	(floating point)
Y location of point #1	(floating point)
Z location of point #1	(floating point)
X location of point #2	(floating point)
Y location of point #2	(floating point)
Z location of point #2 (to 49 control points)	(floating point)
X location of last point	(floating point)
Y location of last point	(floating point)
Z location of last point	(floating point)
End	

3D__Bezier

X location of point #1	(floating point)
Y location of point #1	(floating point)
Z location of point #1	(floating point)
X location of point #2	(floating point)
Y location of point #2	(floating point)
Z location of point #2 (to 12 control points)	(floating point)
X location of last point	(floating point)
Y location of last point	(floating point)
Z location of last point	(floating point)
End	

3D__Face

X location of point #1	(floating point)
Y location of point #1	(floating point)
Z location of point #1	(floating point)
X location of point #2	(floating point)
Y location of point #2	(floating point)
Z location of point #2	(floating point)
X location of point #3	(floating point)
Y location of point #3	(floating point)
Z location of point #3	(floating point)
X location of point #4	(floating point)
Y location of point #4	(floating point)
Z location of point #4	(floating point)
End	

ASCII Codes ... Appendix B

ASCII, an acronym for American Standard Code for Information Interchange, is a commonly accepted system for assigning numbers to printable alphanumeric characters, punctuation and symbols, used in the English language. Each character in a character set or font has its own number.

ASCII codes 0 to 127 represent the same characters across all computer systems. However, since a character set can extend as high as number 255, numbers 128 to 255 often represent unique characters in a particular computer system. The higher numbers can represent line drawing symbols, international characters or even another international character set.

Codes 0 to 31 are reserved for system control codes and such non-printing characters as a carriage return (ASCII 13), line feed (ASCII 10), escape (ASCII 27) or form feed (ASCII 12). They are sometimes used for non-standard and graphics characters. It is unlikely you will need to change or alter these characters. ASCII number "0" is the NUL character and cannot be changed by the vector font editor (note that this is not the character "0" which is ASCII number 48).

Codes 128 to 255 (hex \$80 to \$FF) are usually unique to each computer system, font or even to each computer manufacturer. Characters occupying these positions in the system character set are displayed in the font editor.

When you alter a font, you change the symbol assigned to that ASCII number, but not the actual number or the keyboard assignments which remain the same. Pressing a key on the keyboard prints whatever character is currently assigned to that number. For example, if you change ASCII number 65 "A" to a symbol of a transistor, then when you load that font into DynaCADD and press "A", a transistor symbol is displayed instead.

Since programmers also use hexadecimal numbering systems, the information below is provided with decimal and hexadecimal ASCII numbers, and the character the numbers represent.

Appendix B, ASCII Codes

Dec	Hex	Character	Dec	Hex	Character
0	0	NUL (cannot be changed)	32	20	space
1	1	Ctrl A SOH	33	21	!
2	2	Ctrl B STX	34	22	"
3	3	Ctrl B ETX	35	23	#
4	4	Ctrl B EOT	36	24	\$
5	5	Ctrl E ENQ	37	25	%
6	6	Ctrl F ACK	38	26	&
7	7	Ctrl G Bell	39	27	'
8	8	Ctrl H Backspace	40	28	(
9	9	Ctrl I HT	41	29	)
10	A	Ctrl J Line Feed	42	2A	*
11	B	Ctrl K VT	43	2B	+
12	C	Ctrl L Form Feed	44	2C	,
13	D	Ctrl M Carriage Return	44	2D	-
14	E	Ctrl N SO	46	2E	.
15	F	Ctrl O SI	47	2F	/
16	10	Ctrl P DLE	48	30	0
17	11	Ctrl Q DC1	49	31	1
18	12	Ctrl R DC2	50	32	2
19	13	Ctrl S DC3	51	33	3
20	14	Ctrl T DC4	52	34	4
21	15	Ctrl U NAK	53	35	5
22	16	Ctrl V SYN	54	36	6
23	17	Ctrl W ETB	55	37	7
24	18	Ctrl X CAN	56	38	8
25	19	Ctrl Y EM	57	39	9
26	1A	Ctrl Z SUB	58	3A	:
27	1B	Ctrl [Escape	59	3B	;
28	1C	Ctrl \ FS	60	3C	<
29	1D	Ctrl] GS	61	3D	=
30	1E	Ctrl ^ RS	62	3E	>
31	1F	Ctrl _ US	63	3F	?

Appendix B, ASCII Codes

Dec	Hex	Character
64	40	@
65	41	A
66	42	B
67	43	C
68	44	D
69	45	E
70	46	F
71	47	G
72	48	H
73	49	I
74	4A	J
75	4B	K
76	4C	L
77	4D	M
78	4E	N
79	4F	O
80	50	P
81	51	Q
82	52	R
83	53	S
84	54	T
85	55	U
86	56	V
87	57	W
88	58	X
89	59	Y
90	5A	Z
91	5B	[
92	5C	\
93	5D	]
94	5E	^
95	5F	_

Dec	Hex	Character
96	60	`
97	61	a
98	62	b
99	63	c
100	64	d
101	65	e
102	66	f
103	67	g
104	68	h
105	69	i
106	6A	j
107	6B	k
108	6C	l
109	6D	m
110	6E	n
111	6F	o
112	70	p
113	71	q
114	72	r
115	73	s
116	74	t
117	75	u
118	76	v
119	77	w
120	78	x
121	79	y
122	7A	z
123	7B	{
124	7C	
125	7D	}
126	7E	~
127	7F	DEL

DynaCADD Command List ... Appendix C



Screen Control Commands

		TRANCOPY
		SHOWCOM
		DZEXTENTS
		DZLAST
		DZIN
		DZOUT
		DZALL
		DZWINDOW
		SCROLL
		CENTER
		REPAINT
		REGEN

Insertion and Location Commands

X	Y	Z	PANG
IX	IY	IZ	PRAD
LOC	END	ORG	INT
ON	SET	FILTER	,

Entity Selection Commands

BYDIG	BYALL	BYWIN	BYWOUT
BYVWIN	BYPWIN	BYLAST	BYLAYER
BYCOLOR	BYSTYLE	BYWEIGHT	BYCHAIN
FILTER			

X	Y	Z	PANG
IX	IY	IZ	PRAD
LOC	END	ORG	INT
ON	SET	FILTER	,

BYDIG	BYALL	BYWIN	BYWOUT
BYVWIN	BYPWIN	BYLAST	BYLAYER
BYCOLOR	BYSTYLE	BYWEIGHT	BYCHAIN
FILTER			

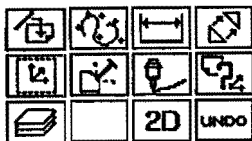
Appendix C, DynaCADD Command List

ICON PAD COMMANDS

VERB

NOUN

MODIFIERS



	INSERT
	CURVE
	DIMENSION
	TRANSFORM
	EDITVIEW
	CREATE
	OUTPUT
	INFORM
	EDITLAYER
	INT2D
	INT3D
	UNDO

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 INSERT		
 +	 POINT	
 ^	 LINE	
 ○	 CIRCLE	
 ⌒	 ARC	
 ⌒	 FILLET	
 ⬭	 ELLIPSE	
 ⬭	 ELLARC	
 □	 BOX	
 ○	 POLYGON	
 A	 TEXT	
 T	 SUBFIG	
 T	 POLYFIG	
 ▨	 SECTION	
 ▩	 HATCH	
 ■	 SOLID	
 □	 FACE	
 ☒	 DELETE	

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 INSERT	 POINT	
	 LINE	
	   	 LINELOC
	   	 ORTHO
	   	 VERTICAL
		 HORIZONTAL
		 PERPEND
		 PARALLEL
		 TANTO
		 TANPOINT
		 LINANG (Flag)
		 LINLEN (Flag)
	 CIRCLE	
	   	 C2POINT
	   	 CIRRAD
		 CIRDIA
		 CIRCIRCUM
		 CBETWEEN
		 CIRRADLOC
	 ARC	
	   	 ARC3POINT
	   	 ARCRAD

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 INSERT	 ARC	 ARCDIA  ARCCIRCUM  FROM ARCSTARTA (Flag)  TO ARCENDA (Flag)  ABETWEEN  ARCRADLOC
	 FILLET    	 FILRAD  FILDIA  CFILRAD  CFILDIA
	 ELLIPSE      ROTN   	 ELLMAJRAD (Flag)  ELLMINRAD (Flag)  ELLMAJDIA (Flag)  ELLMINDIA (Flag)  ROTN  ELLROT (Flag)
	 ELLARC      ROTN  FROM  TO 	 ELMAJRAD (Flag)  ELAMINRAD (Flag)  ELAMAJDIA (Flag)

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 INSERT	 ELLARC	 ELAMINDIA (Flag)
		 ELAROT (Flag)
		 ELASTARTA (Flag)
		 ELAENDA (Flag)
	 BOX	
	 POLYGON	
	 ? SIDES	 ? SIDES POLYSIDES (Flag)
	 ROTN	 ROTN POLYROT (Flag)
		 POLYRAD (Flag)
		 POLYDIA (Flag)
		 POLYAFLAT (Flag)
		
		
		
	 TEXT	
		 TEXTWID (Flag)
		 TEXTHEI (Flag)
	 ROTN	 ROTN TEXTROT (Flag)
		 TEXTSLANT (Flag)
		 SETPSPACE (Flag)
		 TEXTLEFT (Flag)
		 TEXTRIGHT (Flag)
		 TEXTCENT (Flag)
		 TEXTLOC
		 TEXTDIGI
	 TEXT + TEXT	
		
		

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 INSERT	 TEXT	 TEXTASSUM  TEXT2ROT
	 SUBFIG	
	   	 SFIGROT (Flag)  SFIGXSCA (Flag)  SFIGYSCA (Flag)  SFIGZSCA (Flag)
	 POLYFIG	
	   	 PFIGROT (Flag)  PFIGXSCA (Flag)  PFIGYSCA (Flag)  PFIGZSCA (Flag)
	 SECTION	
	   	 SECTDIST (Flag)  SECTANGLE (Flag)
	 HATCH	
	   	 HATCHX (Flag)  HATCHY (Flag)
	 SOLID	
	 FACE	
	 DELETE	

Appendix C, DynaCADD Command List

VERB	NOUN				MODIFIERS	
 CURVE						
 BSPL CURVE	 BEZ CURVE	 CURVE & CURVE		 BSPL CURVE	BSPLINE	
 CURVE	 MOD CURVE			 BEZ CURVE	BEZIER	
				 CURVE CURVE	CONVERT	
				 BSPL BEZ	 BEZ BSPL	
					 BSPL BEZ	TOBEZIER
					 BEZ BSPL	TOBSPLINE
				 BREAK	BREAK	
				 JOIN	JOIN	
				 MOD CURVE	MODCURVE	
				 MOVE CP	 DELETE CP	
				 INSC PB	 INSC PA	
				 REVERSE	REVERSE	

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 DIMENSION		
   	 DIMLOC	
   	   	 TEXTWID (Flag)
   		 TEXTHEI (Flag)
   		 TEXTSLA (Flag)
	 DIMLINE	See DIMLOC
	 DIMHORI	"
	 DIMVERT	"
	 DIMRAD	"
	 DIMDIA	"
	 DIMHCHAIN	"
	 DIMVCHAIN	"
	 CHCHAIN	"
	 CVCHAIN	"
	 DIMHBASE	"
	 DIMVBASE	"
	 DIMCLINE	"
	 DIMCCLINE	"
	 DIMANGLE	"
	 DIMLEADER	"

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 TRANSFORM		
	 DELETE	
	 TRIM	
	 DIVIDE	
	 TRANSLATE	
	 ROTATE	
	 MIRROR	
	 MODIFY	
	 STRETCH	
	 SCALE	
	 OFFSET	
	 EQUIDIST	
	 PROJECT	
	 REVOLVE	
	 SWEEP	
	 ARRAY	
	 MASK	
	 UNMASK	
	 BURNVIEW	
	 ALIGN	

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 TRANSFORM	 DELETE	
	 TRIM	
	   	 TRIMONE  TRIMCORN  TRIMINTOP  TRIMMULTI  TRIMINC  TRIMDIST
	 DIVIDE	
	   	 DIVLOC  DIVNDIV
	 TRANSLATE	
	   	 TRANMOVE  TRANCOPY  TRANMCOPY  TRANSFACE
	 ROTATE	
	   	 ROTANG (Flag)  ROTNCOPY (Flag)
	 MIRROR	

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 TRANSFORM	 MODIFY	
	   	 CHARAD
	   	 CHALOC
	   	 CHAPEN
		 CHALAYER
		 CHATWID
		 CHATHEI
		 CHATROT
		 CHATSLA
		 CHATSPACE
		 CHATFONT
		 EDITTEXT
		 TXTORIENT
	 STRETCH	
	 SCALE	
	 OFFSET	
	   	 THROUGH
	   	 DISTANCE
		 SOLIDPATH
		 TRACEPATH
		 TRACEENDS
	 EQUIDIST	

Appendix C, DynaCADD Command List

VERB



TRANSFORM

NOUN



PROJECT



REVOLVE



SWEEP



ARRAY



MASK



UNMASK



BURNVIEW



ALIGN

MODIFIERS



PROJDEPTH (Flag)



PROJSCALE (Flag)



PROJFACE (Flag)



REVANG (Flag)



REVNCOPY (Flag)



REVN MESH (Flag)



REVFACE (Flag)



SWEEPFACE (Flag)



INCX (Flag)



NX (Flag)



INCY (Flag)



NY (Flag)



INCZ (Flag)



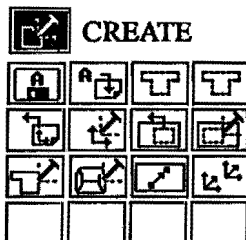
NZ (Flag)

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 EDITVIEW		
   	 DELVIEW	
   	 DEFWIEW	
   	   	 CVIEWGCP (Flag)
   		 CVIEWWCL (Flag)
	 VIEWGCP	
	 ROTVIEW	
	   	 AXROT (Flag)
		 AYROT (Flag)
		 AZROT (Flag)
	 REVCLIP	
	 REVORG	
	 REVSCALE	
	 RESETVIEW	
	 DYNAMIC	
	 VIEWZIN	
	 VIEWZOUT	
	 VIEWZEXT	
	 SCRVIEW	
	 HIDE	
	 SEEK	

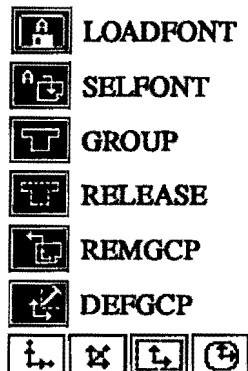
Appendix C, DynaCADD Command List

VERB



CREATE

NOUN



LOADFONT

SELFONT

GROUP

RELEASE

REMGCP

DEFGCP



REMIMAGE

DEFIMAGE

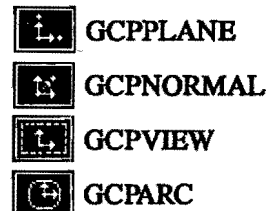
DEFFIG

DEFMACRO

DEFORIGIN

TRANGCP

MODIFIERS



GCPPLANE

GCPNORMAL

GCPVIEW

GCPARC

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 OUTPUT		
   	 PLOTTER	
	 PRINTER	
	 POSTSCRIPT	
	 POINTFILE	
 INFORM		
   	 LIST	
	   	 LSTDRAW
	   	 LSTIMAGE
		 LSTGCP
		 LSTVIEW
		 LSTLAYER
		 LSTEXTENT
		 LSTMACRO
	 COUNT	
	 VERIFY	
	 MEASURE	
	   	 MEASANGLE
		 MEASDIST
		 MEASPERIM
		 MEASAREA

Appendix C, DynaCADD Command List

VERB	NOUN	MODIFIERS
 EDITLAYER		
   	 SEL LAYER	
   	 SEL LAYER T	
   	 LAYER ON	
	 LAYER OFF	
	 LAY LOCK	
	 LAY UNLOCK	
	 LAYER NAME	
	 COPY LAYER	
	 COLLAYER (Flag)	
	 COLSTYLE (Flag)	
	 COLWEIGHT (Flag)	
 INT12D		
 INT13D		
 UNDO		

Appendix C, DynaCADD Command List

Printer and Plotter Support ... Appendix D

The following is a list of the printer and plotter driver files provided with DynaCADD. If your output device is not listed, and is not compatible with any of these devices, please call our technical support number for more information.

Or, if your plotter is not listed here, and you have the proper documentation, we provide a utility called MAKEPLOT to enable driver customization. Simply run the program, load a similar driver and fill in the necessary information in the dialog box in order to create a driver.

fig. (243) MakePlot Dialog

Plotters

907_1023.PLT	Calcomp 907 1023
907_1043.PLT	Calcomp 907 1043
960X1023.PLT	Calcomp 960 1023
960X1043.PLT	Calcomp 960 1043
* CAMM-1.PLT	Roland CAMM-1 sign cutter
CPGL1023.PLT	Calcomp 1023 CPGL
DMPL42.PLT	HI DMP-42
DMPL52.PLT	HI DMP-52
DMPL56.PLT	HI DMP-56
DMPL61.PLT	HI DMP-61
DMPL62.PLT	HI-DMP-62

Appendix D, Printer and Plotter Support

DPX-2200.PLT	Roland DG DPX-2200
DPX-3300.PLT	Roland DG DPX-3300
DRAFTPRO.PLT	HP DraftPro
DRFTMAST.PLT	HP DraftMaster
DXY-990.PLT	Roland DG DXY-990
GEM.PLT	GEM Metafile
GRX-400.PLT	Roland GRX-400
HP-7580.PLT	HP 7580
HP-7585.PLT	HP 7585-7586
IOLN3700.PLT	Ioline LP3700
IOLN4000.PLT	Ioline LP4000
PCI_1023.PLT	Calcomp PCI 1023
PCI_1043.PLT	Calcomp PCI 1043
SP600.PLT	SweetPea 600/601

- * The CAMM-1 is a desktop vinyl cutter for use in creating signs, labels, etc. Special CAMM-1 fonts are available upon request. When using the CAMM-1 make sure that your drawing uses only line weight 1.

Printers

GMNI-15D.PNT	Gemini 15 Draft (9 pin)
IMG072.PNT	GEM bit-image file 72 DPI
IMG150.PNT	GEM bit-image file 150 DPI
IMG300.PNT	GEM bit-image file 300 DPI
LASRJETD.PNT	HP LaserJet 150 DPI
LASRJETF.PNT	HP LaserJet 300 DPI
LQ-1000.PNT	Epson LQ-1000 13.6" sheet (24 pin)
LQ-1000S.PNT	Epson LQ-1000 8" sheet (24 pin)
LQ-2500.PNT	Epson LQ-2500 13.6" sheet (24 pin)
LQ-2500S.PNT	Epson LQ-2500 8" sheet (24 pin)
LQ-500.PNT	Epson LQ-500 8" sheet (24 pin)
LQ-510.PNT	Epson LQ-510 8" sheet (24 pin)
PR-1012D.PNT	Roland DG PR-1012 draft 8" sheet (9 pin)
PR-1012F.PNT	Roland DG PR-1012 final 8" sheet (9 pin)
PR-1012M.PNT	Roland DG PR-1012 medium 8" sheet (9 pin)
PR-1215D.PNT	Roland DG PR-1215 draft 13.6" sheet (9 pin)
PR-1215F.PNT	Roland DG PR-1215 final 13.6" sheet (9 pin)
PR-1215M.PNT	Roland DG PR-1215 medium 13.6" sheet (9 pin)

DynaCADD Files and Folders ... Appendix E

This is a list of all the files provided on the DynaCADD disks.

Disk 1:

INSTALL.EXE	DynaCADD installation utility
README!	Current info on DynaCADD

\GEMAPPS\GEMSYS

GEMVDI.EXE	GEM/3 VDI
GEM.EXE	GEM/3 program file
GEM.RSC	GEM/3 resource file
SDPSC9.VGA	GEM/3 VGA color driver
SDPSM9.VGA	GEM/3 VGA monochrome driver
SDEHF9.EGA	GEM/3 EGA color driver
SDEH19.EGA	GEM/3 EGA monochrome driver
SDHRC9.EGA	GEM/3 Hercules® monochrome driver

\DYNACADD

DYNACADD.APP	DynaCADD program file
--------------	-----------------------

\DYNACADD\RESOURCE

DYNACADD.RSC	DynaCADD resource file
--------------	------------------------

\DYNACADD\CONFIG

DYNACADD.ALT	DynaCADD alerts
ICON.DAT	Icon structure file
ICONS.DAT	Icon bit-image file
HELPMMSG.DAT	On-screen help messages
DEFAULT.DWG	Default (seed) drawing
KEYS.DAT	Key translation file

Disk 2:

\DYNACADD

FONTEEDIT.APP	Font Editor program file
MAKEPLOT.APP	Make Plot program file

Appendix E, DynaCADD Files And Folders

\DYNACADD\RESOURCE

FONTEEDIT.RSC	Font Editor resource file
MAKEPLOT.RSC	Make Plot resource file

\DYNACADD\CONFIG

STANDARD.STY	Default line styles
FONTEEDIT.ALT	Font Editor alerts
HELP.HLP	Online documentation

\DYNACADD\FONTS

HATCH.FNT	Hatch font
LEROY__R.FNT	Leroy Rough
LEROY__S.FNT	Leroy Smooth
ARCHIRE.FNT	AGFA Compugraphic Architecture 1
ARCH2BD.FNT	AGFA Compugraphic Architecture 2
CGTM4RE.FNT	AGFA Compugraphic Times
TRIU4RE.FNT	AGFA Compugraphic Triumvirate
OUTLINE.FNT	Outline (hollow) font
BRUSH.FNT	AGFA Compugraphic Brush
DOMC4RE.FNT	AGFA Compugraphic Dom Casual
PARKAVEN.FNT	AGFA Compugraphic Park Avenue
REVU2LT.FNT	AGFA Compugraphic Revue Light
ZCHN5ME.FNT	AGFA Compugraphic Zapf Chancery
ENGLISH.FNT	English font
OLD__ENGL.FNT	AGFA Compugraphic Old English
FONTEEDIT.FNT	Font Editor default font

\DYNACADD\DRIVERS

Plotter/Printer drivers	See Appendix D
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\DYNACADD\PARTS

DynaCADD 2D and 3D tutorial files

DynaCADD Error Information ... Appendix F

This is not a complete list of DynaCADD messages. Most of the messages are self explanatory and need no further explanation; they are usually simple requests for confirmation or warnings about the result of a continued action. Some messages may require more information; they are explained below. Others are similar to those described below, for example non-intersecting entities.

Many messages are file error messages: invalid file name, the file is the wrong format, the disk is full. Sometimes a DynaCADD file is not found by the program when needed. In these situations you can:

- Select a filename from the File Selector dialog rather than typing one.
- Select another file.
- Load an old format file into an earlier version of DynaCADD and save it as a DEF file. Then load it into this version of DynaCADD through the File Transfer option.
- Use your backup copy of the file if the disk is bad.
- Insert the appropriate backup of your DynaCADD disks into the drive. If the file still is not found, save your part and quit, then copy the missing file(s) to the backup disks from your original disks.
- Use another formatted disk to save the file(s). Disks must have sufficient room for the files.

If you still do not understand a message, call our technical support line for further help and assistance.

Appendix F, DynaCADD Error Information

Saving the current Part will overwrite any existing files. Do you wish to continue with saving the current Part?

You are saving the part with a filename already in use. If you continue, the file on disk is overwritten. Similar warnings are displayed for other file types.

Unable to allocate sufficient memory for another Font definition.

Remove one or more fonts from memory before trying to load a new font.

UNDO will reverse ALL changes made to the database during the last major command. Do you wish to continue?

Undo is a powerful command. Make sure you want to Undo the last command before you continue.

No Font has been selected. Select a Font and retry.

A font must be loaded before you can use any function that involves text, including dimensioning.

Figure not a DynaCADD file.

The file you selected is not a DynaCADD file. If the file loads into an earlier version of DynaCADD, load it into that version, then save it as a DEF file. Use the File Transfer option in the File menu to load it into DynaCADD, then save it again as a figure file.

Figure and Drawing Mode are mismatched.

You cannot insert 2-D figures into 3-D mode and vice versa. Change mode and try again.

Error in database file. File possibly damaged.

The file may be corrupted. Try using a backup copy of the file. If the file loads into an earlier version of DynaCADD, load it into that version, then save it as a DEF file. Use the File Transfer option in the File menu to load it into DynaCADD, then save it again.

Unable to create directory. Part has not been activated.

Select Drawing from the Set menu and activate a part before continuing.

Appendix F, DynaCADD Error Information

A Part must be active before a Drawing may be activated.

Select Drawing from the Set menu and activate a part before continuing.

Activating a new Drawing will abort any changes to the current Drawing which have not been saved.

If you wish to retain your drawing, save the part before activating a new drawing.

No Part currently active. Please activate a Part.

No Drawing currently active. Please activate a Drawing.

You cannot enter CADD level until both a part and a drawing have been activated.

Resultant zero line length disallowed.

The line has no length. Try selecting new locations or use the LINLEN flag.

No Intersection between the selected location and the suggested angle was found.

Try selecting either a different entity or a different location.

The origins of the selected Arcs or Circles are too close to calculate a valid tangent.

Either move one of the entities or select at least one different entity and try again.

Unable to define the plane the 3D Fillet orientation requires.

No Intersection between the selected entities was found. Fillet insertion aborted.

A fillet cannot be placed between entities which do not intersect. Move the entities and try again, or select other entities.

Fillet would result in a selected line being trimmed to zero line length. Fillet insertion aborted.

The resulting line cannot be trimmed to zero length after a fillet is inserted. Choose another location and try again.

Appendix F, DynaCADD Error Information

Font definition not found. The font has not been loaded.

Select a font filename from the file selector dialog.

MEMORY WARNING. The current command cannot be completed due to lack of free memory available

Select Database sort from the File menu, then save your part. Select clear to free memory and then restore to bring back the part.

The selected entities are non contiguous.

Trim the entities to make sure they intersect and form a contiguous group.

Unable to remove the current GCP Definition.

Change the current GCP definition, then try again.

Error: GCP table overflow GCP not added to Drawing.

Error: Image table overflow Image not added to Drawing.

Error: View definition overflow. View not added to Drawing.

There are already the maximum definitions in the tables, delete at least one before trying again.

Warning: GCP Skewed to View!

The GCP is not aligned with the plane of view. Select another GCP and try again.

Unable to allocate a command history window.

The command history has been terminated due to lack of free memory.

You have run out of memory for a command history window. Perform a database sort, then save the part. Use Clear and then Restore and try again.

Item Selector ... Appendix G

The Item Selector dialog appears whenever DynaCADD needs a file name to be entered or selected.

Directory Line

The directory line shows the disk and folders where the items listed in the directory window are located. To change the information on the directory line, place the pointer anywhere on the line and click once. The text

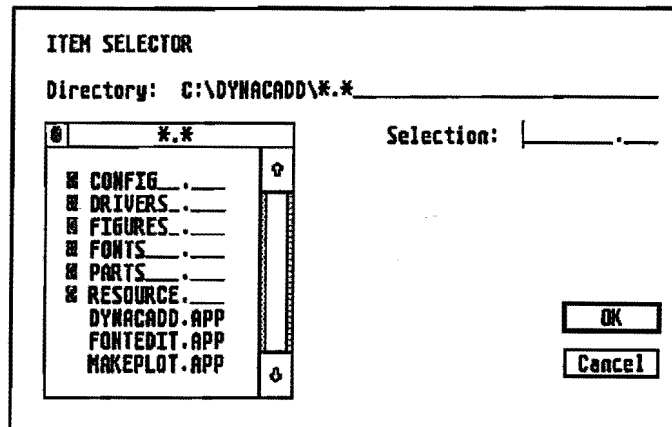


fig. (244) Item Selector Dialog

cursor will appear after the last character on the line. Press the [Backspace] key to erase individual characters or press [Esc] to clear the entire line. See Editing Dialog Text in Chapter 1 for further dialog editing information.

You can enter an asterisk (*) or question marks (?) in the directory line as “wildcard” characters. For example, C:\DYNACADD\PARTS*.PRT means “all DynaCADD part files in a folder called PARTS in the DYNACADD folder on disk drive C”. (See your operating system manual if you need more information about wildcard characters).

After you enter the information on the directory line, click inside the directory window or press [Enter] to update the window with a list of items in that folder.

Appendix G, Item Selector

Selection Line

Information typed on the selection line represents the file name that you wish to use for the subsequent operation. Click on OK or press [Enter] to indicate acceptance of the file name.

Directory Window

The directory window lists the folders and files in the location specified on the directory line. The window can only show nine items at a time. Use the scroll bar, arrows, and slider to move up and down in the folder if it contains more than nine items.

You can use any of the following methods to select an item from the directory window:

- Click on the selection line and the text cursor will appear at the beginning of the line. Type the file name and then click on the OK button or press [Enter]. This method is mostly used to enter in a new file name when saving or writing a file.
- Click on the file name in the directory window and the name will be inserted into the selection line. Click on OK or press [Enter].
- Double-click on the file name in the directory window. This is the fastest method for selecting files.

Folder Indicator

Names listed in the directory window with a diamond shape to the left are folders within the directory path shown on the directory line. Click on the folder name to display its contents in the directory window.

Close Box

The directory window has a close box (a diamond-shaped symbol) in the upper left corner. Click on the close box to move back to the previous folder.

If there is no previous folder, the directory window shows you the drive identifiers for your computer. Click on the drive identifier to display the contents of its main directory in the directory window.

Clicking on Cancel will leave the Item Selector dialog, ignoring any selections or directory changes.

Fonts .. Appendix H

CG Times

33	!	57	9
34	"	58	:
35	#	59	;
36	\$	60	<
37	%	61	=
38	&	62	>
39	'	63	?
40	(	64	@
41	)	65	A
42	*	66	B
43	+	67	C
44	,	68	D
45	-	69	E
46	.	70	F
47	/	71	G
48	0	72	H
49	1	73	I
50	2	74	J
51	3	75	K
52	4	76	L
53	5	77	M
54	6	78	N
55	7	79	O
56	8	80	P

DynaCADD®

81	Q	105	i
82	R	106	j
83	S	107	k
84	T	108	l
85	U	109	m
86	V	110	n
87	W	111	o
88	X	112	p
89	Y	113	q
90	Z	114	r
91	[	115	s
92	\	116	t
93	]	117	u
94	^	118	v
95	—	119	w
96	'	120	x
97	a	121	y
98	b	122	z
99	c	123	{
100	d	124	
101	e	125	}
102	f	126	~
103	g	127	
104	h		

CG Times

128	Ç	154	Ü	180	œ	206	232	★
129	ü	155	ç	181	Œ	207	233	☆
130	é	156	£	182	À	208	234	■
131	â	157		183	Ã	209	235	□
132	ä	158	ß	184	Ö	210	236	◀
133	à	159	f	185	˘	211	237	▶
134	å	160	á	186	˙	212	238	
135	ç	161	í	187	†	213	239	
136	ê	162	ó	188	‡	214	240	
137	ë	163	ú	189	©	215	241	±
138	è	164	ñ	190	®	216	242	
139	ï	165	Ñ	191		217	243	
140	î	166	ª	192	ij	218	244	
141	ì	167	º	193	II	219	245	/
142	Ä	168	¿	194		220	246	÷
143	Å	169	‡	195		221	247	●
144	É	170	”	196		222	248	°
145	æ	171	“	197		223	249	•
146	Æ	172	”	198		224	250	·
147	ô	173	¡	199		225	251	✓
148	ö	174	«	200		226	252	-
149	ò	175	»	201		227	253	-
150	û	176	ã	202		228	254	—
151	ù	177	õ	203		229	255	—
152	ÿ	178	Ø	204		230		
153	Ö	179	ø	205		231		

CG Triumvirate

33	!	57	9
34	"	58	:
35	#	59	;
36	\$	60	<
37	%	61	=
38	&	62	>
39	'	63	?
40	(	64	@
41	)	65	A
42	*	66	B
43	+	67	C
44	,	68	D
45	-	69	E
46	.	70	F
47	/	71	G
48	0	72	H
49	1	73	I
50	2	74	J
51	3	75	K
52	4	76	L
53	5	77	M
54	6	78	N
55	7	79	O
56	8	80	P

DynaCADD®

81	Q	105	i
82	R	106	j
83	S	107	k
84	T	108	l
85	U	109	m
86	V	110	n
87	W	111	o
88	X	112	p
89	Y	113	q
90	Z	114	r
91	[	115	s
92	\	116	t
93	]	117	u
94	`	118	v
95	—	119	w
96	'	120	x
97	a	121	y
98	b	122	z
99	c	123	{
100	d	124	
101	e	125	}
102	f	126	-
103	g	127	
104	h		

CG Triumvirate

128	Ç	154	Ü	180	œ	206	232	★
129	ü	155	Ɔ	181	Œ	207	233	☆
130	é	156	£	182	À	208	234	■
131	â	157		183	Ã	209	235	□
132	ä	158	ß	184	Ö	210	236	◀
133	à	159	f	185	”	211	237	▶
134	å	160	á	186	’	212	238	
135	ç	161	í	187	†	213	239	
136	ê	162	ó	188	‡	214	240	
137	ë	163	ú	189	©	215	241	±
138	è	164	ñ	190	®	216	242	
139	ï	165	Ñ	191		217	243	
140	î	166	ª	192	ij	218	244	
141	ì	167	º	193	Ij	219	245	/
142	Ä	168	¿	194		220	246	÷
143	Å	169	‡	195		221	247	●
144	É	170	”	196		222	248	◦
145	æ	171	“	197		223	249	•
146	Æ	172	”	198		224	250	·
147	ô	173	¡	199		225	251	✓
148	ö	174	«	200		226	252	-
149	ò	175	»	201		227	253	-
150	û	176	ã	202		228	254	—
151	ù	177	õ	203		229	255	—
152	ÿ	178	Ø	204		230		
153	Ö	179	ø	205		231		

Park Avenue

33	!	57	9
34	"	58	:
35	#	59	;
36	\$	60	<
37	%	61	=
38	&	62	>
39	'	63	?
40	(	64	@
41	)	65	A
42	*	66	B
43	+	67	C
44	,	68	D
45	-	69	E
46	.	70	F
47	/	71	G
48	0	72	H
49	1	73	I
50	2	74	J
51	3	75	K
52	4	76	L
53	5	77	M
54	6	78	N
55	7	79	O
56	8	80	P

DynaCADD®

81	Q	105	i
82	R	106	j
83	S	107	k
84	T	108	l
85	U	109	m
86	V	110	n
87	W	111	o
88	X	112	p
89	Y	113	q
90	Z	114	r
91	[	115	s
92	\	116	t
93	]	117	u
94	^	118	v
95	_	119	w
96	`	120	x
97	a	121	y
98	b	122	z
99	c	123	{
100	d	124	
101	e	125	}
102	f	126	-
103	g	127	
104	h		

Park Avenue

128	Ç	154	Ü	180	æ	206	232	★
129	ü	155	€	181	Œ	207	233	☆
130	é	156	£	182	À	208	234	■
131	â	157		183	Ã	209	235	□
132	ã	158	ß	184	Ö	210	236	◀
133	à	159	f	185	-	211	237	▶
134	ä	160	á	186	'	212	238	
135	ç	161	í	187	†	213	239	
136	ê	162	ó	188	‡	214	240	
137	ë	163	ú	189	©	215	241	±
138	è	164	ñ	190	®	216	242	
139	ï	165	Ñ	191		217	243	
140	î	166	ª	192	ÿ	218	244	
141	ì	167	º	193	Œ	219	245	/
142	Ä	168	¿	194		220	246	÷
143	Å	169	‡	195		221	247	●
144	É	170	"	196		222	248	°
145	æ	171	"	197		223	249	•
146	Æ	172	"	198		224	250	·
147	ø	173	¡	199		225	251	✓
148	ö	174	«	200		226	252	-
149	ò	175	»	201		227	253	—
150	û	176	ã	202		228	254	—
151	ù	177	õ	203		229	255	—
152	ÿ	178	Ø	204		230		
153	Ö	179	ø	205		231		

Old English

DynaCADD®

33	!	57	9	81	Q	105	i
34	"	58	:	82	R	106	j
35	#	59	;	83	S	107	k
36	\$	60	<	84	T	108	l
37	%	61	=	85	U	109	m
38	&	62	>	86	V	110	n
39	'	63	?	87	W	111	o
40	(	64	@	88	X	112	p
41	)	65	A	89	Y	113	q
42	*	66	B	90	Z	114	r
43	+	67	C	91	[	115	s
44	,	68	D	92	\	116	t
45	-	69	E	93	]	117	u
46	.	70	F	94	^	118	v
47	/	71	G	95	—	119	w
48	0	72	H	96	'	120	x
49	1	73	I	97	a	121	y
50	2	74	J	98	b	122	z
51	3	75	K	99	c	123	{
52	4	76	L	100	d	124	
53	5	77	M	101	e	125	}
54	6	78	N	102	f	126	~
55	7	79	O	103	g	127	
56	8	80	P	104	h		

Old English

128	Q	154	ſ	180	æ	206	232	★
129	ü	155	†	181	œ	207	233	☆
130	é	156	£	182	Ä	208	234	■
131	â	157		183	Ã	209	235	□
132	ä	158	ß	184	Ö	210	236	◀
133	à	159	f	185	..	211	237	▶
134	ä	160	á	186	'	212	238	
135	ç	161	í	187	†	213	239	
136	ê	162	ó	188	¶	214	240	
137	ë	163	ú	189	©	215	241	±
138	è	164	ü	190	®	216	242	
139	ï	165	Ń	191		217	243	
140	î	166	"	192	ij	218	244	
141	ì	167	"	193	II	219	245	/
142	Ä	168	ç	194		220	246	÷
143	Å	169	‡	195		221	247	●
144	Æ	170	"	196		222	248	○
145	æ	171	"	197		223	249	●
146	Æ	172	„	198		224	250	•
147	ô	173	¡	199		225	251	✓
148	ö	174	«	200		226	252	~
149	ò	175	»	201		227	253	-
150	û	176	ā	202		228	254	—
151	ù	177	ō	203		229	255	—
152	ÿ	178	Œ	204		230		
153	Ŧ	179	ſ	205		231		

Dom Casual

33	!	57	9
34	"	58	:
35	#	59	;
36	\$	60	<
37	%	61	=
38	&	62	'
39	'	63	?
40	(	64	@
41	)	65	A
42	*	66	B
43	+	67	C
44	,	68	D
45	-	69	E
46	.	70	F
47	/	71	G
48	0	72	H
49	1	73	I
50	2	74	J
51	3	75	K
52	4	76	L
53	5	77	M
54	6	78	N
55	7	79	O
56	8	80	P

DynaCADD®

81	Q	105	i
82	R	106	j
83	S	107	k
84	T	108	l
85	U	109	m
86	V	110	n
87	W	111	o
88	X	112	p
89	Y	113	q
90	Z	114	r
91	[	115	s
92	\	116	t
93	]	117	u
94	^	118	v
95	_	119	w
96	`	120	x
97	a	121	y
98	b	122	z
99	c	123	{
100	d	124	
101	e	125	}
102	f	126	~
103	g	127	
104	h		

Dom Casual

128	Ç	154	Ü	179	■	205	231	
129	Û	155	†	180	œ	206	232	★
130	ó	156	£	181	Æ	207	233	☆
131	â	157		182	À	208	234	■
132	ä	158	ß	183	Ã	209	235	□
133	à	159	f	184	Ö	210	236	◀
134	á	160	á	185	˜	211	237	▶
135	ç	161	í	186	'	212	238	
136	ê	162	ó	187	†	213	239	
137	ë	163	ú	188	¶	214	240	
138	è	164	ñ	189	©	215	241	±
139	ï	165	Ñ	190	®	216	242	
140	î	166	ª	191		217	243	
141	ì	167	º	192	ij	218	244	
142	Ä	168	¿	193	U	219	245	/
143	Å	169	‡	194		220	246	÷
144	É	170	"	195		221	247	●
145	æ	171	"	196		222	248	°
146	Æ	172	"	197		223	249	•
147	ô	173	¡	198		224	250	•
148	ö	174	«	199		225	251	✓
149	ò	175	»	200		226	252	-
150	û	176	ã	201		227	253	-
151	ù	177	õ	202		228	254	-
152	ÿ	178	Ø	203		229	255	-
153	Ö			204		230		

Zapf Chancery

DynaCADD®

33	!	57	9	81	Q	105	i
34	"	58	:	82	R	106	j
35	#	59	;	83	S	107	k
36	\$	60	<	84	T	108	l
37	%	61	=	85	U	109	m
38	&	62	>	86	V	110	n
39	'	63	?	87	W	111	o
40	(	64	@	88	X	112	p
41	)	65	A	89	Y	113	q
42	*	66	B	90	Z	114	r
43	+	67	C	91	[	115	s
44	,	68	D	92	\	116	t
45	-	69	E	93	]	117	u
46	.	70	F	94	^	118	v
47	/	71	G	95	—	119	w
48	0	72	H	96	'	120	x
49	1	73	I	97	a	121	y
50	2	74	J	98	b	122	z
51	3	75	K	99	c	123	{
52	4	76	L	100	d	124	
53	5	77	M	101	e	125	}
54	6	78	N	102	f	126	~
55	7	79	O	103	g	127	
56	8	80	P	104	h		

Zapf Chancery

128	Ç	154	Û	180	œ	206	232	★
129	ü	155	¢	181	Œ	207	233	☆
130	é	156	£	182	À	208	234	■
131	â	157		183	Ã	209	235	□
132	ã	158	ß	184	Ö	210	236	◀
133	à	159	f	185	˘	211	237	▶
134	å	160	á	186	'	212	238	
135	ç	161	í	187	†	213	239	
136	ê	162	ó	188	¶	214	240	
137	ë	163	ú	189	©	215	241	±
138	è	164	ñ	190	®	216	242	
139	ï	165	Ñ	191		217	243	
140	î	166	ª	192	ij	218	244	
141	ì	167	º	193	Ij	219	245	/
142	Ä	168	¿	194		220	246	÷
143	Å	169	‡	195		221	247	●
144	É	170	"	196		222	248	°
145	æ	171	"	197		223	249	•
146	Æ	172	"	198		224	250	•
147	ô	173	¡	199		225	251	✓
148	ö	174	«	200		226	252	-
149	ò	175	»	201		227	253	-
150	û	176	ã	202		228	254	—
151	ù	177	õ	203		229	255	—
152	ÿ	178	Ø	204		230		
153	Ö	179	ø	205		231		

ARCHITECTURE NO.1

DynaCADD®

!"#\$%&'()*+,-./0123456789:;<=>?@
 ABCDEFGHIJKLMNOPQRSTUVWXYZ[
 \]^_`'ABCDEFGHIJKLMNPOQRSTUVWXYZ
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Glossary of Terms ... Appendix I

- 2-D:** Two dimensional locations without depth or space, which can be defined using a X-Y coordinate system. Entities with only width and height. See also Page origin.
- 3-D:** Three dimensional locations which can be specified using an X-Y-Z coordinate system. Entities with width, height and depth. See also GCP.
- Absolute Coordinates:** Used by the coordinate system to determine the origin of the current part or drawing.
- Acute Angle:** An angle less than 90 degrees. See also Obtuse Angle.
- Allowance:** The intentional difference in the dimensions of mating parts that determines the class of fit; an intentional difference to determine clearance (positive allowance) or interference (negative allowance). See Tolerance.
- Arc:** A segment of a circle. It has an origin (the center of a circle), start and end angles.
- Array:** The collection of entities or groups of entities into an ordered series, either linear or circular. The TRANSFORM ARRAY command creates linear arrays. Circular arrays are created with the TRANSFORM ROTATE and TRANSFORM REVOLVE commands.
- Ascender:** The upper part of lowercase characters such as "b", "d" and "f" which reach above the x-line to the ascent line.
- Ascent Line:** The maximum height to which ascenders may rise above the baseline. Uppercase letters usually only rise to the capline, a small amount lower than the ascent line.
- ASCII:** Acronym for American Standard Code for Information Interchange. This is a commonly accepted system for assigning numbers to printable alphanumeric characters, punctuation and symbols, used by the English language. Each character in a character set has its own number.

Appendix I, Glossary of Terms

Assembly Drawing:	Drawing showing a product in its completed state. See Detail Drawing.
Auxiliary View:	An additional view of an object, usually of a surface inclined to the primary surfaces of the object so that the inclined surfaces can be shown without distortion.
Auto Tolerance:	Automatic display of linear and angular tolerance values in a dimension. There are three auto tolerance styles available.
Axis:	An imaginary line which defines the direction and magnitude of a series of locations; line about which a object rotates, thereby defining a plane. In the Set menu, Axis refers to a series of evenly spaced lines that can be displayed to aid location and entity alignment, similar to the grid, but without the snap feature. Axes is the plural of axis. There are two axes in 2-D: X and Y, and three axes in 3-D: X, Y and Z.
Axometric Projection:	A projected view in which the planes of sight are perpendicular to the plane of projection, but in which the three faces of a rectangular object are all inclined to the plane of projection.
Baseline:	The line on which the base or foot of most characters rests. The baseline is also used to calculate vertical distances between lines.
Baseline Dimension:	A system of dimensioning whereby as many features as practically possible are located from a common datum point. Also called "common point" or "datum" dimensioning. The dimension text is aligned with the defined baseline, either vertically or horizontally.
Basic Dimension:	The theoretical exact size of an object's attribute.
Baud Rate:	The speed at which the computer and output device are communicating through the serial port. Most plotters communicate at 9600 baud.

Bézier Curve:	A curve generated along the path of four or more locations. The fit to the control points is less pronounced than with B-spline curves.
Bit Mapped:	Fonts that are mapped to screen pixels and thus are dependent on the screen resolution for their smoothness and clarity. DynaCADD uses vector fonts, which are always the same at every resolution.
B-spline Curve:	A curve generated along the path of three or more locations. The fit to the control points is more pronounced than with Bézier curves.
Buffer:	A reserved area of memory used to temporarily store information such as entities to copy, font vectors, or the screen.
CADD:	Acronym for Computer Aided Design and Drafting.
CADD Level:	DynaCADD level where you design and modify parts.
Capline:	The line to which the tallest uppercase letters reach. This is often a little lower than the ascent line.
Cartesian Coordinates:	Method of plotting an exact position along the X and Y (and Z in 3-D mode) axes relative to the zero point, in the current drawing units. Can be either relative or absolute.
Center Line:	Line style used to indicate the center points of circles, arcs and polygons, axes of cylindrical parts, or axes of symmetry.
Chain Dimension:	Successive dimensions that extend from one feature to another, on a point to point basis.
Chamfer:	A bevel on an external edge or corner, usually dimensioned by giving the angle and length.
Character:	A single element of a font or character set. This can be any letter, number, punctuation or symbol.

Appendix I, Glossary of Terms

Character Set:	A collection of 255 elements, or characters. Not all of the 255 elements need be complete; there can be any number of blanks. A character set is also one part of a font family and is often referred to by itself as a font.
Chordal Dimension:	Used to specify the spacing of points on a circle relative to a particular datum point or points.
Circular Surface:	Surfaces that have diameter or radius.
Click:	Pressing the mouse button. This is the left mouse button, unless otherwise stated.
Command History:	Display of all user entered commands, information, measurement results, in a scrolling text window. Press [Tab] to view and hide this window.
Control Point:	A location on a curve or at the ends of font vectors, which defines the location of curves and lines. Control points can be moved, deleted or inserted.
Coordinate:	Each part of a system of magnitudes used to fix a position of a location or an entity. See X-Y axes, X-Y-Z axes, GCP, Page Origin.
Contiguous:	Connected, unbroken; uninterrupted in series or sequence.
Crosshair:	Set of intersecting lines showing the current position of the graphics cursor. Used to position points or locations. See also Trap.
Database:	The collection of related information, stored in an predefined, organized format.
Default:	Initial setting of a definable parameter. For example, page size.
Degree:	Unit of angular measurement defined as 1/360 of the circumference of a circle. The 0/360 location in DynaCADD is to the right, or east position, with 90 degrees at the top, or north, increasing in a counter clockwise direction.

Appendix I, Glossary of Terms

Delimiter:	A terminating character specifying the end of a set of characters.
Descender:	The "tail" of lowercase letters such as "g", "j" and "y", that reaches below the baseline to the descent line.
Descent Line:	The lower limit to which a descender can reach.
Detail Drawing:	Drawing showing details of each part of an object or mechanism. See Assembly Drawing.
Deviation:	The total variance between two points or locations.
Dialog:	Easy to use graphics interface designed to display and communicate information between the user and the computer. They appear only when certain commands call them.
Diameter:	The length of a straight line from any point on the circumference, through the origin of a circle, to the opposite point on the circumference. If the entity is an arc, then the diameter is measured as if the arc formed a complete circle.
Digitize:	The selection of a location or entity by means of a light pen, mouse or other input devices; specifying locations by positioning the crosshair and clicking the left mouse button. See Free Digitizing.
Directory:	Part of a pathname used to specify which route the program is to follow to find a particular file. Convenient method of grouping related disk files into a single location.
Drawing:	Two dimensional database created in DynaCADD. This may also be used to reference the physical page on which all designs are displayed.
Drop Down Menu:	Also called pull down menu. Easy to use graphics interface designed to organize related command options or features in a minimal amount of space. Menu titles are displayed on the menu bar, across the top of the screen.

Appendix I, Glossary of Terms

Dynamic Calculator:	The graphic calculator for changing the rotation, scale and zoom factor of 3-D views, as you click on the direction arrows.
Ellipse:	Plane curve made by a point moving so that the sum of the distances from any point on the curve to two fixed points (called foci) is constant.
Enclosed:	Contiguous entities which completely enclose a space, with no gaps, for inserting hatching or section lines.
Entity:	Single base geometric figure, combined together to create larger more complex figures. The basic design element in DynaCADD.
Extension Lines:	Lines that extend perpendicular to a dimension line, indicating the point of the distance or object to which the dimension applies. Also called witness lines.
Extents:	The total area or space used by the drawing or by entity definitions. The database extents are the furthest limits of the entities in every direction. In 3-D mode, this is an imaginary cube that extends along all three axes.
Extrude:	Thrust out from; to project a plane along an axis.
Figure:	A combination of selected entities into a single grouped entity. Often saved and later inserted into another drawing. A subfigure remains its identity as a group when inserted into a drawing. See also Polyfigure.
Fillet:	A concave intersection between two lines or planes. In DynaCADD, a fillet is a special type of arc entity and where ever an arc is a valid entity, so is a fillet. In manufacturing, a fillet is additional material allowed in the intersection of two surfaces.
Filter:	Design tool used to screen selection of entities based on entity type.
Flag:	An optional modifier that affects the function of a related parent command. Angle of rotation, distance, number of polygon sides are all examples of flags.

Appendix I, Glossary of Terms

Folder:	A directory or subdirectory on a disk, used to organize files. When seen as an icon, it is represented by a small graphic of a file folder.
Font:	A set of characters in a particular style, often, but not limited to, text. Hatch patterns are an example of a non text font.
Free Digitizing:	This occurs when you select a location in a view where the GCP is not aligned with the view and DynaCADD cannot tell where the location should go. With entity snap on, there is no free digitizing. See Digitize.
GCP:	Acronym for Geometric Coordinate Plane; the origin of the 3- D coordinate system; the origin of a 3-D part.
GEM:	Acronym for Graphics Environment Manager, the graphic interface created by Digital Research.
Grid:	Regularly spaced locations to which entities and locations can be snapped. The grid locations in DynaCADD are visible as small dots. See also Axis.
Group:	Entities combined as a single, complex entity. All the entities in the group are affected by any command applied to any part of the group, such as translate or rotate.
Half Section:	View of an object, usually symmetrical, showing one half of the view in section. See Sectional View.
Hatch:	Character set used to create regular surface textures in contiguous entities. Often used as terrain or material type symbols or to indicate knurls.
Helix:	A curve generated by moving a point uniformly around and up or down a cylinder. The point must always be parallel to the axis of the cylinder. Plural is helices.
Helpline:	Helplines are user set guides for character construction in the font editor.

Appendix I, Glossary of Terms

Hidden Line:	A line consisting of short, evenly spaced dashes, used to indicate the hidden features of an object.
Hidden Surface:	Surfaces which are hidden in one or more reference planes.
Highlight:	To display in reverse text (ie; white on black).
Icon:	Graphic representation of a command, displayed on the left side of the drawing area, or above the drawing area. Icons simplify the command entry process.
Icon Pad:	Section of the DynaCADD interface used to group related icons. Icons are displayed in a vertical hierarchy.
Image:	Stored definition of a area and zoom value on the drawing page that may be restored when required.
Inclined:	A line or plane at an angle to a horizontal line or plane.
Inclined Surface:	A flat surface inclined in one plane and parallel to the other two planes.
Initialize:	To reset to the original state.
Integrate:	Combine parts to make a complete whole.
Interface:	The portion of a program that acts as an interpreter between computer and user.
Intersection:	The exact location at which entities physically or theoretically cross or meet.
Isometric:	A plane rotated by equal angles about all three axes. Usually the angle is 45 degrees, so that the front corner is toward the viewer.
Justification:	Used to specify how a text block is positioned relative to a selected location; left, right, or centered.
Kerning:	Kerning is a method of reducing the space between printed or displayed characters to bring them together for a more "natural" appearance and to reduce the whitespace between them.

Laser:	Acronym for Light Amplification by Stimulated Emission of Radiation. Laser printers are a recent form of high quality output devices.
Layer:	Design tool used to organize related entities in a drawing onto computer controlled imaginary sheets. DynaCADD supports up to 256 layers, numbered 0 to 255. Layers can also be named. See Chapter 17.
Leader:	Arrows connected by a series of straight lines to direct notes, dimensions, symbols, part or item numbers, and so on, to features on the drawing.
Limits:	The largest and smallest permissible sizes of an object. Also called "limits of size". See also Tolerance.
Line of Intersection:	Line common to two surfaces where both meet.
Macro:	The definition of a series of keystrokes assigned to a single key or key combination. For example, [Ctrl-A].
Major Axis:	The greatest straight line distance between the origin and any point on the circumference of the ellipse. Also called "major diameter".
Mask:	To hide entities from view.
Menu:	See Drop Down Menu.
Menu Pad:	See Icon Pad.
Mesh:	A series of circular rings created around the revolved entities, perpendicular to the axis of revolution.
Minor Axis:	The smallest straight line distance between the origin and any point on the circumference of the ellipse. Also called "minor diameter".
Mode:	Term defining the current state of the DynaCADD design session. Used to determine if user actions are to be interpreted to modify and create two dimensional or three dimensional entities.

Appendix I, Glossary of Terms

Normal:	See Perpendicular.
Oblique Surface:	A surface sloped so that it is not perpendicular to any of the three viewing planes. See Inclined Surface.
Obtuse Angle:	An angle greater than 90 degrees. See Acute Angle.
Origin:	Every entity has a point called its origin, located at the center of that entity. The page origin is the 2-D zero point. The 3-D origin is the zero point of the current GCP.
Orthographic Projection:	A 2-D picture plane formed by perpendicular projectors from the object, to the picture plane. An orthographic view is what you get when looking directly at an object's side or face.
Page Origin:	The origin of the 2-D coordinate system; the X0-Y0 position.
Parallel:	Having the same direction, such as two lines which, if extended, would never meet.
Parallel Surface:	Flat surfaces which are parallel to the three viewing planes.
Parameter:	Mathematical term; quantity which may vary for different cases.
Part:	Three dimensional database created in DynaCADD, represented on the screen by a particular object.
Perimeter:	Length of the outline or circumference of a closed figure.
Perpendicular:	Lines or planes at right angles to a given line or plane. Also called "normal".
Phantom Line:	Line style usually used to indicate hidden or interior shapes. Also used to show mating parts in an assembly drawing.
Plane:	Imaginary surface on which points, lines, or arcs lie.

Appendix I, Glossary of Terms

Polar Coordinates:	System defining locations as a specified angle and distance from the origin. Can be either relative or absolute. DynaCADD defines positive angular increments to be in the counter clockwise direction.
Polar Coordinate Dimensioning:	Indicating the position of an entity by means of a linear dimension and an angle other than 90 degrees (implied by vertical and horizontal center lines in the part).
Polyfigure:	A group of entities saved as a figure file on disk. Unlike a subfigure, a polyfigure breaks into its component entities when inserted into a drawing.
Polygon:	Multi sided regular geometric figure such as a hexagon or pentagon, bounded by straight lines of equal length and containing angles of equal size.
Polyline:	A continuous set of lines sharing a common endpoint.
Polywindow:	An irregularly shaped window created out of polylines.
Pop Up Calculator:	See Value Calculator.
Precision:	The number of positions following the decimal place in a floating point number to be displayed or used for calculations. DynaCADD works in twenty point precision, but you can only set dimensions to display up to nine points.
Primary:	First or original; first rank in a series.
Project:	To extend from one location to another.
Radian:	A unit of angular measurement. 360 degrees equals 6.283185 radians.
Radius:	One half the diameter; the distance between the origin of a circle or arc, and any point on the circumference. Plural is radii.
Relative Coordinates:	Incremental coordinate system which refer to the current or last specified location as the origin.

Appendix I, Glossary of Terms

Round:	Also called a radius, in manufacturing a round is material added to the outside of a piece of material to improve its appearance or to avoid forming a sharp edge. Similar to a fillet on the external side.
Sans Serif:	A font without serifs. Usually used for headlines, titles, etc..
Scale:	Relative dimensions; the ratio of uniform reduction or enlargement commonly used in drafting to accurately represent large areas.
Section:	Regularly spaced lines generated between contiguous and completely enclosed entities. Often used to indicate cylindrical intersections, and to emphasize cross section displays or sectional views. Usually thin and drawn at 45 degrees to the major outline.
Sectional View:	Objects shown with interior detail. Exposed or cut surfaces are displayed with hatch patterns or section lines. See Half Section.
Select:	Choosing a menu option, clicking a dialog button.
Serif:	A font with serifs small "feet" that project from the bottom of the characters. the serif provides a visual continuity for reading, so serif typefaces are usually used for bodies of text.
Skewed:	Slanted or distorted; not at right angles to the plane.
Snap:	Forced alignment to a grid intersection.
String:	In DynaCADD, a string is synonymous with a character array; any collection of characters in a single line.
Subfigure:	See Figure.
System level:	DynaCADD level at which part and drawing management are done and where system parameters are established.
Tangent:	Meeting of a line or surface at any point but not intersecting it; straight line to a curve at any point.

Appendix I, Glossary of Terms

Toggle:	A switch used to change between two states, normally on or off.
Tolerance:	The total permissible amount of variation from the design size or location of a part from that shown in a drawing. There are two types of tolerance: geometric, which specifies deviation from given dimensions, and positional, in which the dimensions are exact but the location of the part displays a permissible deviation. See Auto Tolerance, Limits, Allowance.
Tracking:	Method by which the position of the crosshair is displayed at the upper right of the drawing area: relative or absolute, Cartesian or polar.
Trap:	The small boxes around the crosshair when selecting an entity, used to determine which entity to choose. If an entity lies within the small trap, then it is selected, otherwise DynaCADD moves to the larger trap to see if an entity is within it. The size of the traps are adjusted automatically to suit the view and drawing scales. The size of the traps is displayed with the Inform List Lstdraw command.
Undo:	The ability to restore the drawing to the situation prior to the last change. Available as a command icon or by pressing [Undo].
Unit:	Individual quantity chosen as a standard. For example inches or centimeters; distance defined as a single component.
Unmask:	To restore to view previously hidden (masked) entities.
Value Calculator:	The pop up scientific calculator that appears when a value entry is required.
Vector:	Quantity considered to have length and direction, but no fixed location in space. The position of one location relative to another. In the font editor, a vector is any contiguous collection of control points used to define the lines of a character.

Appendix I, Glossary of Terms

View Clip:	Rectangular area used to confine an orthographic projection of the current part.
View:	Rectangular window used to observe three dimensional entities at a specified scale and rotation. You can have four simultaneous views in DynaCADD. At least one view must be created to insert and modify 3-D entities.
Window:	Rectangular area defined by a set of diagonally opposing 2-D locations.
X-height:	The x-height, or body size, is the vertical size of the lowercase letter "x", measured from the baseline. This is the most important area of any letterform for 90% of the lowercase characters, since it conveys the impression of the size of a letter.
X-line:	The user set helpline line marking the limit of the x height of a character.
X Y Axes:	2-D system of coordinates. The X axis normally runs left to right in ascending values, the Y axis normally runs bottom to top in ascending values. The page origin is at locations X0, Y0.
X Y Z Axes:	3-D system of coordinates. The Z axis runs perpendicular to the X Y axes and defines "depth" of an object. The GCP indicates the X Y Z orientation of the view.
Zero Point:	Position where coordinate axes cross and all axes values are zero: (X0, Y0) in 2-D or (X0, Y0, Z0) in 3-D.
Zoom:	Increasing or decreasing the magnification of the drawing area or just a part of the drawing.

Particulars to the Amiga Version ... Appendix J

Backing up DynaCADD

For your convenience, we have not copy-protected the DynaCADD disks. However, a program key is supplied with DynaCADD. This key is required to operate the program. You must insert the key BEFORE you turn on your computer. If you insert the key while the power is turned on, you may damage both the key and your computer. We allow our registered owners to backup DynaCADD under the terms and conditions specified in the License Agreement located at the back of the manual. Please backup the DynaCADD disks now and place the originals in a safe place. Refer to your operating system manual for exact instructions on how to format and copy disks.

Installing DynaCADD - AMIGA Version

DynaCADD requires at minimum an Amiga with 2 Mb of RAM and a hard disk to operate properly. If your system contains a 68020/68030 microprocessor and either a 68881 or 68882 math co-processor, the InstallHD program will copy a version of DynaCADD to your system that takes full advantage of these processors.

Since a screen resolution of at least 640 by 400 is required, we strongly recommend the Commodore 2320 de-interlacer or the MicroWay Flicker Fixer for use with DynaCADD on machines that do not have these capabilities built in.

To install DynaCADD:

1. With your computer off, insert the DynaCADD key into the joystick port and then turn your computer on.
2. Insert the DynaCADD 1 disk into any drive.
3. Double-click on the DynaCADD 1 icon.
4. Double-click on the InstallHD icon.
5. The InstallHD program will run and ask for the name of the destination hard drive or partition. After having entered a valid name, press RETURN or click on OK.

Appendix J, Particulars to the Amiga Version

6. At this point enter in your name and company into the dialog.
Click on OK to continue.
7. Install will now proceed to copy DynaCADD to your hard disk.
Insert the appropriate disks as requested.

Loading DynaCADD

1. If your computer is turned off, make sure the DynaCADD program key is inserted. If it is not, shut your system off now and insert the key, then turn your system back on.
2. Locate the DynaCADD drawer on your hard disk and open it.
3. Click on the DynaCADD icon. This will run DynaCADD and bring you to the DynaCADD system level.
4. Select Activate Part. Choose a part (.PRT) in the parts sub-directory.
5. Select Activate Drawing. Choose a drawing (.DWG).
6. Click on OK. This will now bring you to the CADD level.

The first time that you run DynaCADD, you enter the program at the System level (see the next chapter). After that, DynaCADD keeps track of where you left off the last time a drawing was saved. When DynaCADD is loaded again a dialog appears. You can choose to return to the previous session by clicking on AUTO, start at the System level by clicking on NO, or quit the program by clicking on EXIT.

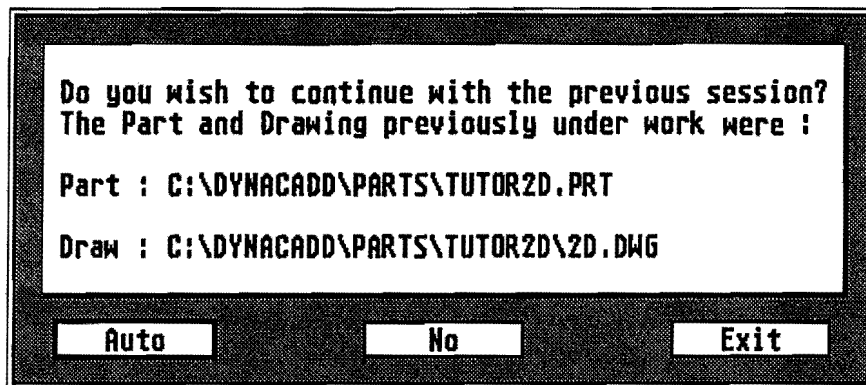


fig. (4) DynaCADD® Auto Start Dialog

Keyboard Conventions

Keyboard conventions differ according to hardware platform. Throughout the manual you will see references to [Undo]. Press and hold either [ALT] or [Amiga] key and then the [U] key. [Enter] and [Return] are interchangeable.

Editing Dialog Text

In DynaCADD there are several dialog boxes where you enter or edit text. Use the following keys for editing:

- | | |
|-----------------|--|
| [Right-Amiga X] | Clears the entry. |
| [Right-Amiga Q] | Performs an undo on edited text. |
| [Backspace] | Erases the character to the left of the cursor. |
| [Delete] | Erases the character to the right of the cursor. |
| [Arrow keys] | Move the cursor in the indicated direction. |
| [Enter] | Accepts the entry and closes the dialog box. |

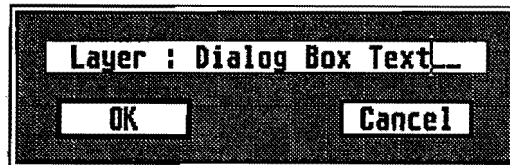


fig. (5) Editing Dialog Text

The Text Editor

The text editor is a basic utility in which you can enter and edit text for several commands. The maximum size of text allowed is 800 characters or one window. The keys used in the text editor for editing are:

[Backspace]	Erases the character to the left of the cursor.
[Delete]	Erases the character to the right of the cursor.
[Up arrow]	Moves the cursor to the end of the previous line.
[Down arrow]	Moves the cursor to the start of the next line.
[Left arrow]	Moves the cursor one character to the left.
[Right arrow]	Moves the cursor one character to the right.
[Enter]	Inserts a carriage return; ends the line and moves the cursor to the start of the line below.
[Esc]	Cancels the editing and returns to the drawing.
[Ctrl-RETURN]	Accepts the entry and closes the text editor.
[Help]	Displays help and information about the text editor.
%%nnn	This allows you to enter special characters which cannot be typed directly or easily through the keyboard. Type %% followed by the three digit number of the special character. The character number must be padded with zeros and have a value between 1 and 255. For example: the hundred and thirty-fourth character of a font is inserted by typing %%134.

Appendix J, Particulars to the Amiga Version

The following special characters used in dimensioning can be entered using a short hand form:

diameter sign %%c

degree sign %%d

plus/minus sign %%p

For more information and a list of special characters see Appendix H.

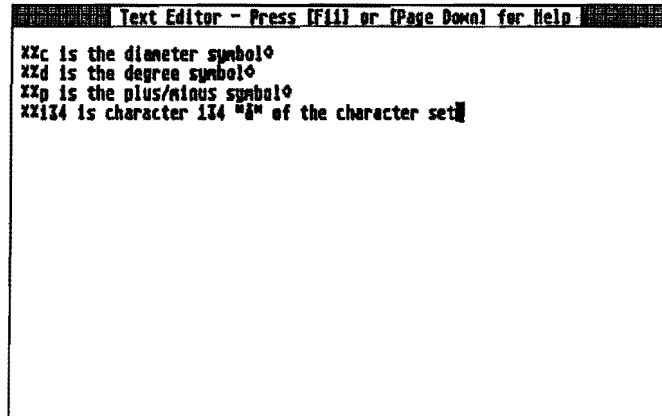
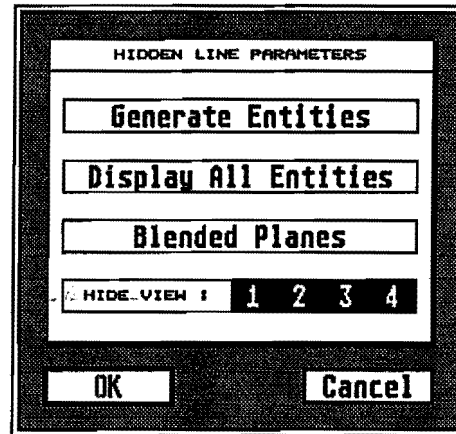
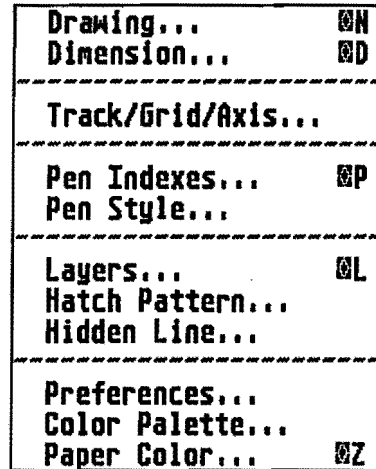


fig. (6) Text Editor

Set Menu

The Amiga version of DynaCADD includes additional functionality; Hidden Line Removal. When Hidden Line is selected from the Set Menu, a dialog appears allowing you to control the parameters of this function.

Set



Hidden Line

This dialog controls how hidden line removal is handled. The 3-D entities in the drawing will remain unchanged by this operation. Additional 2-D entities will be created if the Generate Entities option is enabled. The options to this command include the following:

Generate Entities

Create 2-D lines from the edges of the 3-D faces after the hidden portions of the edges have been removed. The lines are created with the following attributes:

- Layer: On the currently active layer.
- Weight: Pen weight 1.
- Style: Continuous.
- Color: If inheritance is active the line is created using the color of the face, otherwise the currently active color is used.

Appendix J, Particulars to the Amiga Version

Show All Entities

Display all entities after the hidden lines have been generated.

Blended Planes

If two or more 3-D faces share a common edge and the faces are on the same plane, the common edge will be removed and the two faces blended into one. Only the hidden line is changed. The defining faces remain the same.

Hide Views 1-2-3-4

Only the selected views will have the hidden lines removed. A selected view is highlighted. To change the selection status of a view, move the pointer over the appropriate view number and click the left mouse button. The view numbers represent the order at which the views were originally created. The first view created has a view number of 1.

Remove Hidden Lines



Command: EDITVIEW HIDEFACE



Modifiers: None

Mode: 3-D only

Description: HIDEFACE removes all hidden lines. This function is governed by the above mentioned Hidden Line Parameters dialog.

Preferences

The Preferences dialog allows the choice of which device name and number DynaCADD will use when serial or parallel is selected in other dialogs, the speed and method of handshaking, and the number of colors that DynaCADD can display. Clicking on Save stores the settings as the new defaults for future sessions.

Serial Device

The name and unit number of the device that is to be used for serial output can be specified here. This allows the use of multiple serial port cards and other devices for output. Three speeds are available; 9600, 19200 and 38400 baud. All plotters and printers support 9600 baud, some offer the higher rates for decreased output times. Depending on the plotter type or plotter settings, hardware or RTS/CTS handshaking must be used. If RTS/CTS is off, DynaCADD uses software, or X-ON/X-OFF handshaking. Consult your plotter manual for details on the plotter's requirements.

Parallel Device

The name and unit number of the device that is to be used for parallel output can be specified here. This allows output to more devices than just the Amiga's built-in parallel port. Turning fast mode on can significantly reduce the time to output a drawing, however, some printers don't support fast mode and end up printing random characters. If this occurs, simply turn fast mode off.

Screen Setup

Depending on the processor speed and memory configuration of your Amiga, selecting fewer colors can significantly increase the speed of redraws and calculations. For Amiga 2000 based systems we recommend 4 or 8 colors, Amiga 3000 based systems can use 16 colors without too much speed loss. DynaCADD defaults to 4 colors when first installed. The color selection becomes active after clicking on Save, exiting and running DynaCADD.

Appendix J, Particulars to the Amiga Version

Particulars to the ST/TT Version ... Appendix K

Backing up DynaCADD

For your convenience, we have not copy-protected the DynaCADD disks. However, a program key is supplied with DynaCADD. This key is required to operate the program. You must insert the key BEFORE you turn on your computer. If you insert the key while the power is turned on, you may damage both the key and your computer. We allow our registered owners to backup DynaCADD under the terms and conditions specified in the License Agreement located at the back of the manual. Please backup the DynaCADD disks now and place the originals in a safe place. Refer to your operating system manual for exact instructions on how to format and copy disks.

Installing DynaCADD - ST/TT Version

DynaCADD requires at minimum a ST or TT system with 1 Mb of RAM and a hard disk to operate properly. A math co-processor and 2 Mb of RAM is highly recommended. DynaCADD for the Atari includes two specific versions:

1. ST with or without a math co-processor.
2. TT or ST with a 68030 accelerator card with a 68881 math co-processor.

Depending on your system configuration, choose the appropriate DynaCADD version during installation. The ST version of DynaCADD will automatically detect a math co-processor and take full advantage of it. A screen resolution of at least 640 by 400 is required.

To install DynaCADD:

1. With your computer off, insert the DynaCADD key into the joystick port and then turn your computer on.
2. Insert Disk 1 into your computer.
3. Double-click on the Drive icon.
4. Double-click on the HINSTALL.PRГ icon.

Appendix K, Particulars to the ST/TT Version

5. When asked source drive A or B, type the appropriate letter.
6. When asked which version you require, type "S" for ST, "T" for TT or "B" for both.
7. When asked which partition you require, type the appropriate letter.
8. When asked for a specific disk, insert the appropriate disk and press RETURN. The contents of each disk will now be copied to your hard disk.

Loading DynaCADD

Before loading, DynaCADD needs to be fully installed. Please see Installing DynaCADD if DynaCADD has not yet been installed on your system.

To load DynaCADD:

1. Double click on (open) the disk drive icon where the Program is located.
2. If DynaCADD is installed in a folder, double click on (open) the folder within which the Program is located.
3. Double click on the DynaCADD program icon. This brings you to the System level.
4. Select Activate Part. Choose a part (.PRT) in the parts folder.
5. Select Activate Drawing. Choose a drawing (.DWG).
6. Click on OK. This will now bring you to the CADD level.

The first time that you run DynaCADD, you enter the program at the System level. After that, DynaCADD keeps track of where you left off the last time a drawing was saved. When DynaCADD is loaded again a dialog appears. You can choose to return to the previous session by clicking on AUTO, start at the System level by clicking on NO, or quit the program by clicking on EXIT.

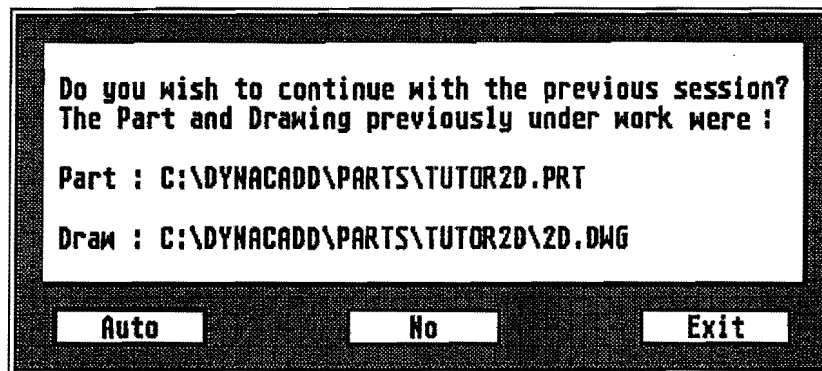


fig. (6) DynaCADD® Auto Start Dialog

The Text Editor

The text editor is a basic utility in which you can enter and edit text for several commands. The maximum size of text allowed is 800 characters or one window. The keys used in the text editor for editing are:

[Backspace]	Erases the character to the left of the cursor.
[Delete]	Erases the character to the right of the cursor.
[Up arrow]	Moves the cursor to the end of the previous line.
[Down arrow]	Moves the cursor to the start of the next line.
[Left arrow]	Moves the cursor one character to the left.
[Right arrow]	Moves the cursor one character to the right.
[Clr/Home]	The first press moves the cursor to the start of the line. The second press, or if the cursor was already at the start of the line, moves the cursor to the start of the text.
[Shift- [Clr/Home]	The first press moves the cursor to the end of the line. The second press, or if the cursor was already at the end of the line, moves the cursor to the end of the text.
[Return]	Inserts a carriage return, ends the line and moves the cursor to the start of the line below.
[Undo]	Cancels the editing and returns to the drawing.
[Enter]	(On the numeric keypad) Accepts the entry and closes the text editor.
[Help]	Displays help and information about the text editor.
%%nnn	This allows you to enter special characters which cannot be typed directly or easily through the keyboard. Type %% followed by the three digit number of the special character. The character number must be padded with zeros and have a value between 1 and 255. For example: the hundred and thirty-fourth character of a font is inserted by typing %%134.

Appendix K, Particulars to the ST/TT Version

The following special characters used in dimensioning can be entered using a short hand form:

diameter sign %%c

degree sign %%d

plus/minus sign %%p

For more information and a list of special characters see Appendix H.

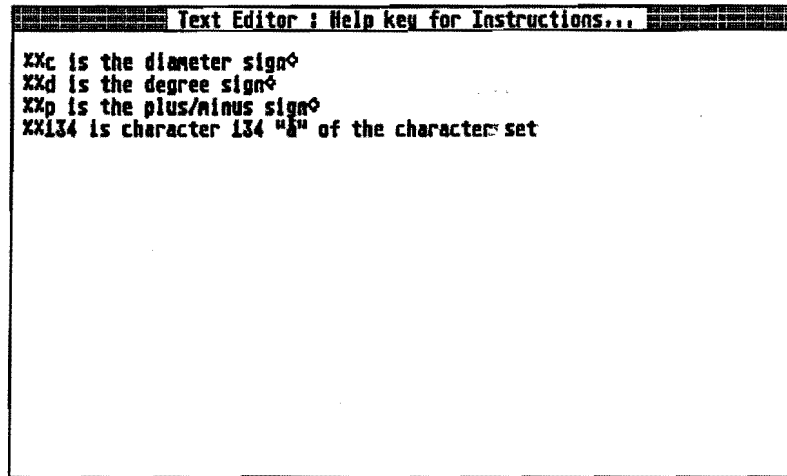
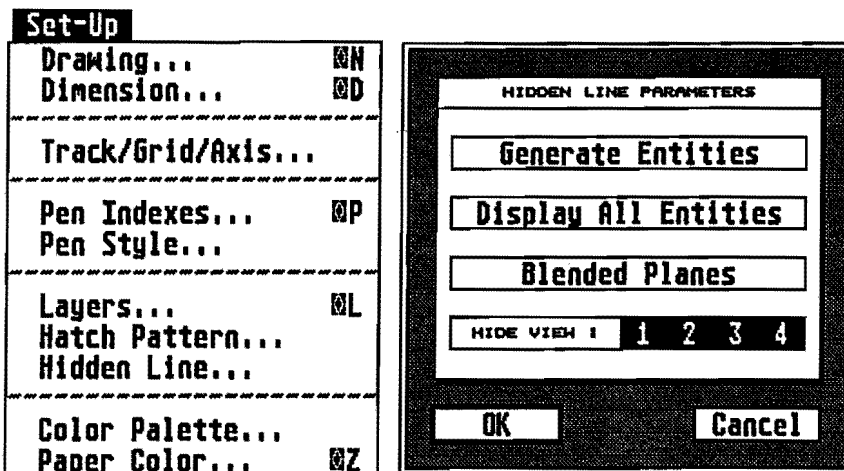


fig. (7) Text Editor

Set-Up Menu

The Atari version of DynaCADD includes additional functionality; Hidden Line Removal. When Hidden Line is selected from the Set Menu, a dialog appears allowing you to control the parameters of this function.



Hidden Line

This dialog controls how hidden line removal is handled. The 3-D entities in the drawing will remain unchanged by this operation. Additional 2-D entities will be created if the Generate Entities option is enabled. The options to this command include the following:

Generate Entities

Create 2-D lines from the edges of the 3-D faces after the hidden portions of the edges have been removed. The lines are created with the following attributes:

- Layer: On the currently active layer.
- Weight: Pen weight 1.
- Style: Continuous.
- Color: If inheritance is active the line is created using the color of the face, otherwise the currently active color is used.

Show All Entities

Display all entities after the hidden lines have been generated.

Blended Planes

If two or more 3-D faces share a common edge and the faces are on the same plane, the common edge will be removed and the two faces blended into one. Only the hidden line is changed. The defining faces remain the same.

Hide Views 1-2-3-4

Only the selected views will have the hidden lines removed. A selected view is highlighted. To change the selection status of a view, move the pointer over the appropriate view number and click the left mouse button. The view numbers represent the order at which the views were originally created. The first view created has a view number of 1.

Remove Hidden Lines



Command: EDITVIEW HIDEFACE



Modifiers: None

Mode: 3-D only

Description: HIDEFACE removes all hidden lines. This function is governed by the above mentioned Hidden Line Parameters dialog.

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1. The first step in the process of identifying a problem is to define the problem. This involves identifying the symptoms of the problem and determining the scope of the problem. Once the problem has been defined, the next step is to identify the causes of the problem. This involves identifying the factors that are contributing to the problem and determining the root cause of the problem. Once the causes of the problem have been identified, the next step is to develop a plan of action to address the problem. This involves identifying the steps that need to be taken to solve the problem and determining the resources that will be needed to implement the plan. Once a plan of action has been developed, the next step is to implement the plan. This involves carrying out the steps that have been identified in the plan and monitoring the progress of the implementation. Finally, the last step in the process is to evaluate the results of the implementation. This involves determining whether the problem has been solved and whether the resources have been used effectively.

2. The second step in the process of identifying a problem is to identify the causes of the problem. This involves identifying the factors that are contributing to the problem and determining the root cause of the problem. Once the causes of the problem have been identified, the next step is to develop a plan of action to address the problem. This involves identifying the steps that need to be taken to solve the problem and determining the resources that will be needed to implement the plan. Once a plan of action has been developed, the next step is to implement the plan. This involves carrying out the steps that have been identified in the plan and monitoring the progress of the implementation. Finally, the last step in the process is to evaluate the results of the implementation. This involves determining whether the problem has been solved and whether the resources have been used effectively.

3. The third step in the process of identifying a problem is to develop a plan of action to address the problem. This involves identifying the steps that need to be taken to solve the problem and determining the resources that will be needed to implement the plan. Once a plan of action has been developed, the next step is to implement the plan. This involves carrying out the steps that have been identified in the plan and monitoring the progress of the implementation. Finally, the last step in the process is to evaluate the results of the implementation. This involves determining whether the problem has been solved and whether the resources have been used effectively.

4. The fourth step in the process of identifying a problem is to implement the plan. This involves carrying out the steps that have been identified in the plan and monitoring the progress of the implementation. Finally, the last step in the process is to evaluate the results of the implementation. This involves determining whether the problem has been solved and whether the resources have been used effectively.

5. The fifth step in the process of identifying a problem is to evaluate the results of the implementation. This involves determining whether the problem has been solved and whether the resources have been used effectively.

ACTIV

6. The sixth step in the process of identifying a problem is to provide feedback to the individuals and groups involved in the process. This involves identifying the strengths and weaknesses of the process and providing feedback to the individuals and groups involved in the process. This feedback should be provided in a constructive manner and should be used to improve the process for the future.

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